Carlson Agstar

Carlson Software Inc.

User's manual

July 23, 2012

Contents

Chapter 1. Product Overview **1**

Product Overview . . . 2
Quickstart Guide . . . 2
System Requirements . . . 3
Installing Agstar . . . 4
Command Entry . . . 9
Startup Wizard . . . 10
AuthorizingAgstar . . . 12
License Agreement . . . 14
Technical Support . . . 17

Chapter 2. File Commands **19**

New . . . 20
Open . . . 21
Close . . . 21
Save . . . 21
Save As . . . 21
Plot . . . 22
Import LandXML File . . . 26
Import Google Earth File . . . 28
Import/Export Topcon TIN File . . . 29
Import/Export Trimble TTM File . . . 29
Write Polyline File . . . 29
Export LandXML File . . . 31
Export Google Earth File . . . 33
Exit . . . 34

Chapter 3. Edit Commands **35**

Undo . . . 36
Erase . . . 36
Delete Layer . . . 36
Move . . . 36
Copy . . . 37
Explode . . . 37
Offset . . . 38

Trim . . . 38
Scale . . . 38
Extend To Edge . . . 39
Extend by Distance . . . 39
Break by Closed Polyline . . . 40
Break at Intersection . . . 41
Change Properties . . . 41
Change Elevations . . . 42
Rotate . . . 42
Edit Text . . . 44
Text Style . . . 44
Text EnlargeReduce . . . 44
Join Nearest . . . 44
Offset 3D Polyline . . . 45
Entities to Polylines . . . 46
Reverse Polyline . . . 47
Reduce Polyline Vertices . . . 47
Smooth Polyline . . . 47
Add Polyline Vertex . . . 48
Close Polylines . . . 48
Edit Polyline Vertex . . . 48
Open Polylines . . . 49
Remove Polyline Arcs . . . 49
Remove Polyline Segment . . . 49
Remove Polyline Vertex . . . 50
Image Clip . . . 50
Image Frame . . . 51
Image Adjust . . . 51

Chapter 4. View Commands **53**

Zoom - Window . . . 54
Zoom Previous . . . 54
Zoom Center . . . 54
Zoom Extents . . . 54
Zoom IN . . . 54
Zoom OUT . . . 54
Pan . . . 55
3D Viewer Window . . . 55
Regen . . . 56

Twist Screen Standard . . . 56
Twist Screen Line . . . 57
Twist Screen Surveyor . . . 57
Restore Due North . . . 57
Layer ID . . . 57
Change Layer . . . 57
Freeze Layer . . . 58
Isolate Layer . . . 58
Restore Layer . . . 58
Thaw Layer . . . 58
List . . . 58
Polyline Info . . . 59
Drawing Inspector . . . 59

Chapter 5. Draw Commands 61

Line . . . 62
2D Polyline . . . 62
3D Polyline . . . 63
Circle . . . 64
Insert . . . 64
Text . . . 65
Insert Symbols . . . 66
Shrink-Wrap Entities . . . 67
3 Point Curve . . . 68
PC PT Radius Point . . . 68
PC Radius Chord . . . 68
Raster Image . . . 69
Place Image by World File . . . 71
Place Google Earth Image . . . 71
Custom Linework Label Formatter . . . 73
Draw Barscale . . . 75
Draw North Arrow . . . 75

Chapter 6. Settings Commands 77

Drawing Setup . . . 78
Units Control . . . 79
Object Snap . . . 81
Set Environment Variables . . . 84
Toolbars . . . 86

Options . . . 87
Edit Symbol Library . . . 100
Configure . . . 101

Chapter 7. Survey Commands **109**

Configure Survey . . . 110
Equipment Commands . . . 118
CSI GBX Pro . . . 118
Geodimeter . . . 118
Impulse Laser . . . 120
Leica GPS System 500 . . . 120
Leica TC Series . . . 122
Manual Total Station . . . 123
Navcom Configuration Guide . . . 123
Navcom GPS Setup . . . 125
Nikon Total Stations . . . 129
Simulation GPS . . . 130
Sokkia . . . 130
Topcon Total Stations . . . 132
Trimble . . . 134
Align GPS To Local Coordinates . . . 137
Align to Benchmark . . . 140
Typical Alignment Scenarios . . . 140
Survey Benchmark . . . 141
Survey Perimeter . . . 142
Survey Interior Surface . . . 143
Survey Subdivision line . . . 144
Survey Master Benchmark . . . 145
Monitor GPS Position . . . 146
Stakeout Design Surface . . . 147

Chapter 8. Design Commands **148**

Set Perimeter . . . 149
Create Existing Ground Grid . . . 149
Design Field . . . 149
Adjust Field Elevation . . . 151
Draw Subdivision Line . . . 151
Assign Subdivision Area Name . . . 152
Area Name Inspector . . . 153

Label Area Names . . . 154
Surface Inspector . . . 154

Chapter 9. Tools Commands 156

Follow Elevation . . . 157
Distance MeasureSwathing . . . 158
Distance Measure with Pen . . . 160
Stakeout Design Surface . . . 160
Watershed Analysis . . . 161
Run Off Tracking . . . 167
Point Defaults . . . 167
DrawLocate Points . . . 169
Draw Field to Finish . . . 172
List Points . . . 208
Edit Points . . . 208
Erase Points . . . 209
Import TextASCII File . . . 209
Export TextASCII File . . . 210
Set Coordinate File . . . 212
CooRDinate File Utilities . . . 212
Create Design Surface Points . . . 217
Edit Point Attributes . . . 218
Move Point Attributes . . . 219
Resize Point Attributes . . . 219
Erase Point Attributes . . . 219
Inverse . . . 220
Occupy Point . . . 220
Traverse . . . 221
Side Shots . . . 222
EnterAssign Point . . . 223
Area Defaults . . . 223
Inverse with Area . . . 227
Area by Interior Point . . . 229
Area by Closed Polylines . . . 229
BearingBearing Intersect . . . 231
BearingDistance Intersect . . . 231
DistanceDistance Intersect . . . 232
EditProcess Raw File . . . 232

Chapter 10. Display Commands **243**

Display . . . 244

CutFill Spreadsheet . . . 244

Pad Design Report . . . 246

Display Options . . . 247

Chapter 11. GIS Commands **249**

GIS Database Settings . . . 250

Define GIS Features . . . 250

Input-Edit GIS Data . . . 252

GIS Inspector . . . 253

GIS Inspector Settings . . . 254

GIS Query/Report . . . 255

Import SHP File . . . 257

Export SHP File . . . 259

Chapter 12. Help Commands **261**

OnLine Help . . . 262

Training Movies . . . 262

Product Overview 1

This chapter describes the product overview, and system requirements for operating Agstar. It also contains instructions for installing and authorizing Agstar, setting up your first project, and a description of non-menu specific commands.

Product Overview

Quickstart Guide

Startup Wizard

Command Entry

System Requirements

Installing Agstar

Authorizing Agstar

Technical Support

Product Overview

Agstar offers a complete package of land-levelling tools for agricultural and irrigation purposes. It carries a versatile suite of commands designed to simplify and speed up the professional surveyor's work, as well as reduce costs by minimizing the amount of dirt to move.

For data collection, Agstar supports a wide range of GPS RTK receivers, including such popular brands as Navcom, Ashtech, Leica, Sokkiam, and Trimble; if you have it, Agstar probably supports it, which means you may not even need to upgrade your hardware.

Agstar handles both the survey and the design sides of land-levelling, allowing you to survey large fields with GPS, and then determine the optimal field design requiring the least amount of dirt moved. Agstar enables you to set the cut/fill ratio, the amount of dirt to import or export, and a host of other design parameters. It also allows you to subdivide the field and assign different designs to each subdivision, which is useful if you want crowns in your field.

After field design, Agstar can display design information in a variety of ways:

1) A pad report detailing the total cubic cut/fill yardage, the field area, the field grade, and other useful information.

2) A color-coded cut/fill spreadsheet (cutsheet) showing the amount of cut/fill for each field segment.

3) A color-coded contour map depicting the existing or cut/fill contour lines.

4) An option to "Stakeout Design Surface", which tells the user the cut or fill with an on-screen lightbar, while navigating the field.

Agstar is the latest in a long line of powerful software survey tools produced by Carlson Software, including Carlson SurvCE and Carlson Survey. In addition to it's basic land survey and design commands, it contains many of the useful features of it's predecessors, allowing you to view and manipulate your survey data in a wide variety of ways.

Carlson Software also provides some of the best technical support in the industry. If you ever need help with it, don't hesitate to contact us for free technical support.

Quickstart Guide

You will find all of the most commonly used Agstar command under the *Survey*, *Design*, and *Display* menus.

To survey and design your first field, it is recommended that you consult the following sections of the manual (in order):

1) *Survey/Configure Survey* - In this section, you must set your *Equipment type* to the GPS equipment you are using. Then in the *GPS Settings* menu, set your *Project Type* to *State Plane 83*, and set your *Zone* to correspond to the local region in which you are working.

2) *Survey/Equipment Setup* - In this section, you must set up your equipment specific setting. This usual requires configuring your stationary GPS unit to base, and your mobile GPS unit to rover.

3) *Survey/Survey Master Benchmark* - In this section, you will survey a reference point, defining the origin of the coordinate system.

4) *Survey/Survey Perimeter* - In this section, you will survey the border of your field.

5) *Survey/Survey Interior Surface* - In this section, you will survey the interior of your field.

6) *Design/Create Existing Ground Grid* - In this section, you will define a grid interval, a necessary step when your survey is complete, and you want to begin designing the field.

7) *Design/Design Field* - In this section, you will generate a design for you field.

8) Now use any of the command under the *Display* menu to generate design information. To inspect the field design with your mouse, run *Design/Surface Inspector*. To view the cuts and fills at your vehicle's position using an onscreen lightbar, go to *Tools/Stakeout Design Surface*.

To create field subdivisions, consult the following commands (in order):

1) *Survey/Survey Subdivision Line* or *Design/Draw Subdivision Line* - The first of these allows you to create a subdivision line with your vehicle. The second allows you to create a subdivision line by drawing it with the mouse.

2) *Design/Assign Subdivision Area Names* - Use this command to name or rename a subdivision area.

3) *Design/Area Name Inspector* - Use this command to view existing field area names.

System Requirements

Operating System

Microsoft® Windows® 98, Windows Millennium Edition (ME), Windows XP Professional, Windows 2000 Professional, or Windows® NT 4.0 with SP 6.0 or later.

Notes: It is recommended that you install and run Agstar on an English version of the operating system. Users of Windows NT 4.0 or Windows 2000 Professional must have Administrator permissions to install Agstar. Not assigning these permissions can cause Agstar to perform incorrectly. See Windows Help for information about assigning user permissions.

Processor

Intel® Pentium® III, IV or AMD-K6® III PC, 450MHz or higher

RAM

128 MB

Video

VGA display 1024 x 768

Hard disk

500MB free disk space

Pointing device

Mouse

CD-ROM

Any speed (for installation only)

GPS Equipment

RTK base and rover

Optional hardware

Printer or plotter

Digitizer

Modem or access to an Internet connection

Open GL-compatible 3D video card

The OpenGL driver that comes with the 3D graphics card must have the following: Full support of OpenGL or later. An OpenGL Installable Client Driver (ICD). The graphics card must have an ICD in its OpenGL driver software. The "miniGL" driver provided with some cards is not sufficient for use with this Autodesk CAD engine.

Web browser

Microsoft Internet Explorer 5.0

Netscape Navigator 4.5 or later

Installing Agstar

If you're installing Agstar on Microsoft® Windows NT® 4.0 and Windows 2000, you must have permission to write to the necessary system registry sections. To do this, make sure that you have administrative permissions on the computer on which you're installing.

Before you install Agstar, close all running applications. Make sure you disable any virus-checking software. Please refer to your virus software documentation for instructions.

Note: If you are upgrading from an older version of Agstar, you must uninstall the older version before installing Agstar. This is required for successful software installation and to meet the guidelines of the EULA (End User License Agreement).

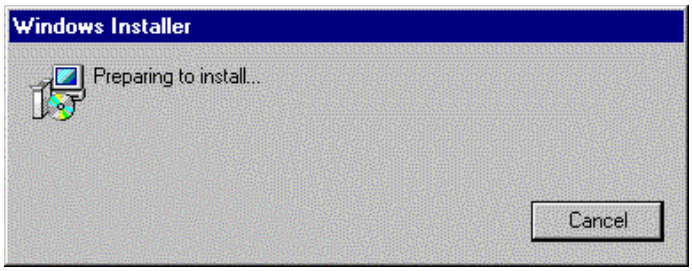
1 Insert the CD into the CD-ROM drive.

If Autorun is enabled, it begins the setup process when you insert the CD.

To stop Autorun from starting the installation process automatically, hold down the SHIFT key when you insert the CD.

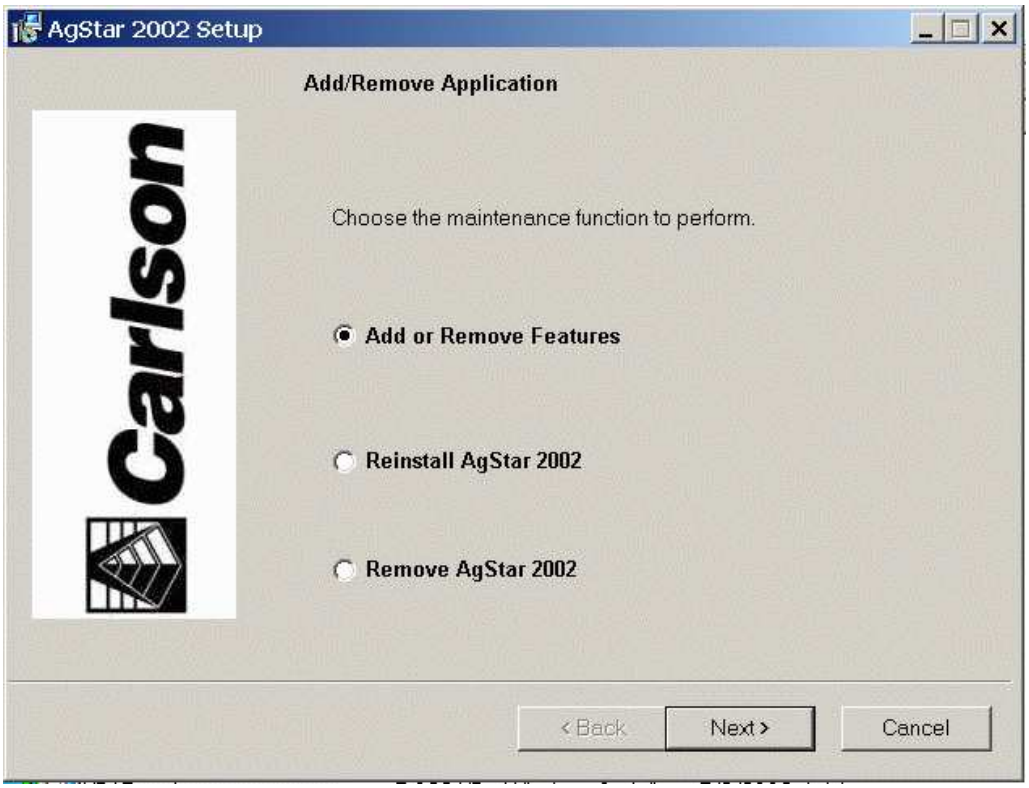
To start the installation process without using Autorun, from the Start menu (Windows), choose Run. Enter the CD-ROM drive letter, and setup. For example, enter d:\setup.

2 The Windows Installer dialog box is displayed.



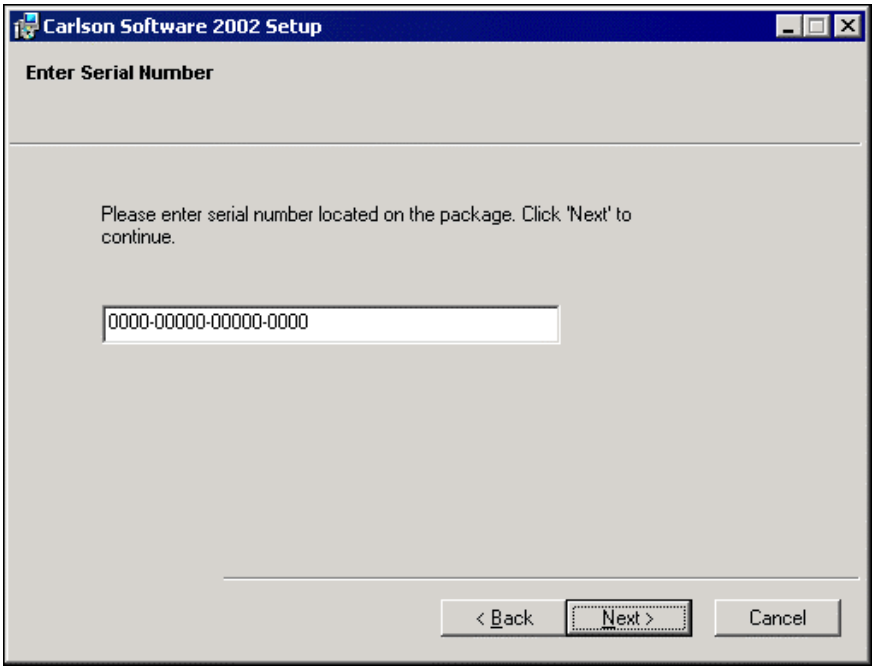
3 After reading the initial Agstar dialog box, press Next. If this is the initial installation, you will see the dialog shown below.

If this version of Agstar has already been installed, you will see the slightly different dialog shown below.



In this case, it is recommended that you remove the current installation. After the current installation is removed, you may start the install process once more to continue.

4 On the Serial Number dialog box, you must enter the serial number provided with your copy of Agstar.



5 On the Select Installation Type dialog box, select the type of installation you want: Typical, Compact, or Custom. Choose Next.

Typical installs the following features:

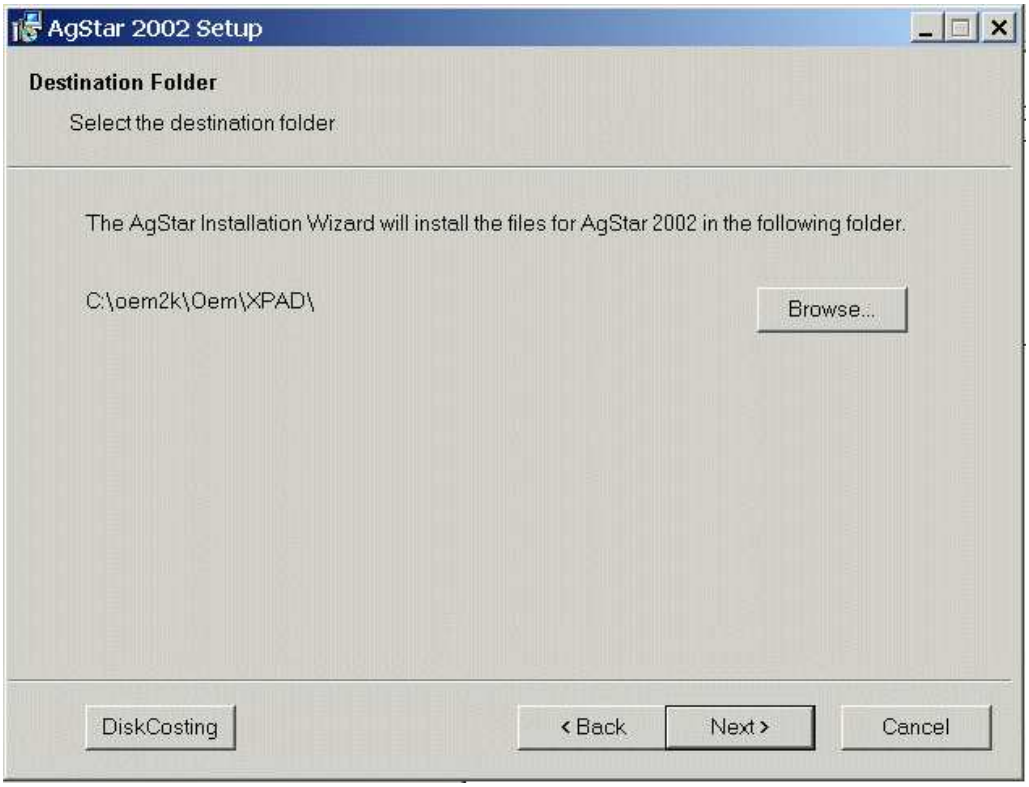
- Program files: Executables, menus, toolbars, Help templates, TrueType® fonts, and additional support files
- Internet tools: Support files
- Fonts: SHX fonts
- Samples: Sample drawings
- Help files: Online documentation

Compact installs only the program files and fonts.

Custom installs only the files you select. By default, the Custom installation option installs all Agstar features. To install only the features you want, choose a feature, and then select one of the following options from the list:

- Will be installed on local hard drive: Installs a feature or component of a feature on your hard drive.
- Entire feature will be installed on local hard drive: Installs a feature and its components on your hard drive.
- Feature will be installed when required only: Installs a feature on demand.
- Entire feature will be unavailable: Makes the feature unavailable.

6 On the Destination Folder dialog box, do one of the following:

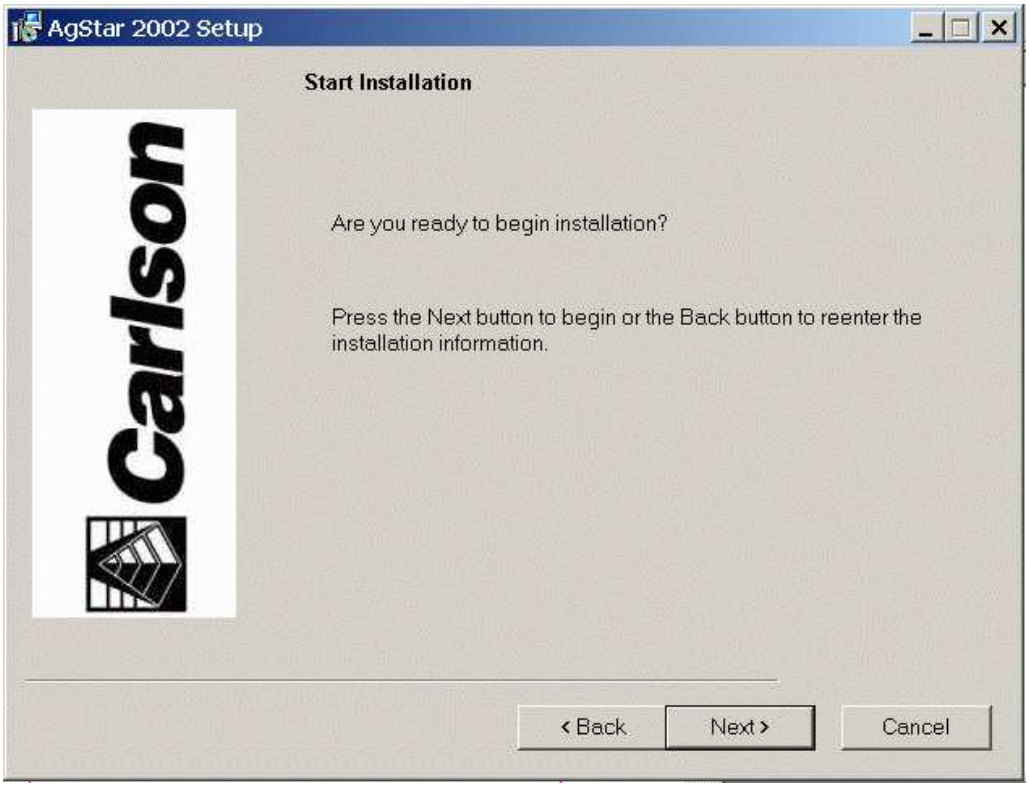


Choose Next to accept the default destination folder/directory.

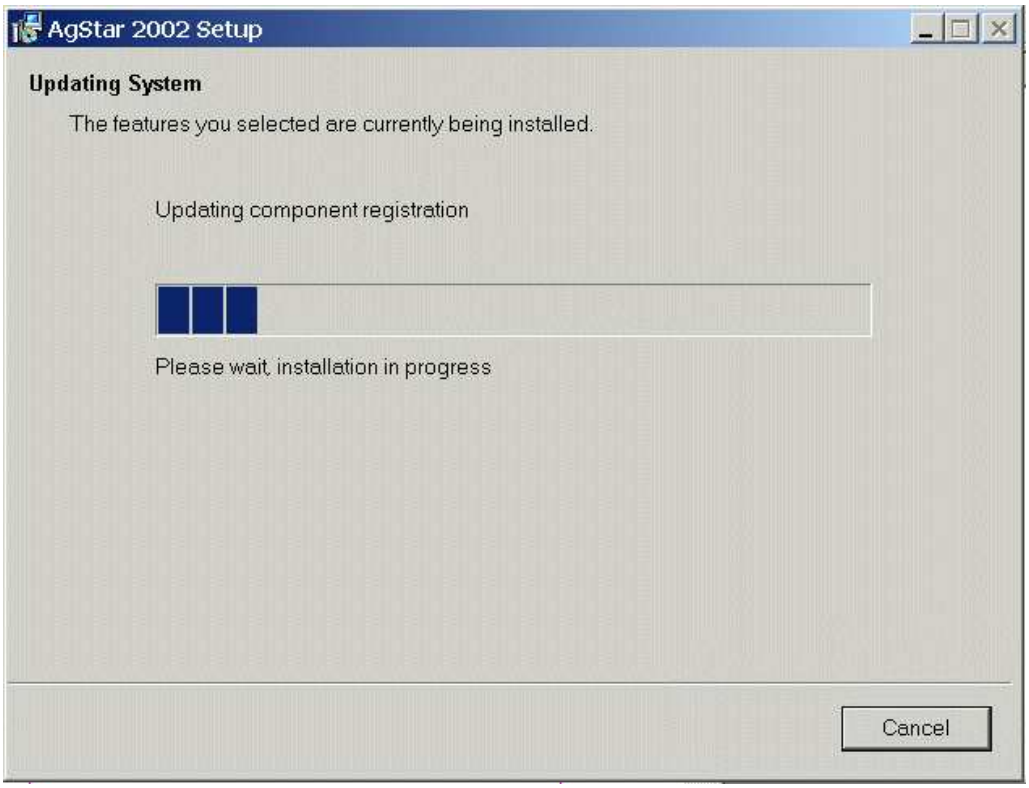
Choose Browse to specify a different drive and folder where you want Agstar to be installed. Choose any directory that is mapped to your computer (including network directories) or enter a new path. Choose OK and then Next.

Setup installs some files required by Agstar in your system folder (for example, c:\Windows\System, or c:\Winnt\System32). This folder may be on a different drive than the folder you specify as the installation folder (for example, d:\Program Files\ Agstar). You may need up to 60 MB of space in your system folder, depending on the components you select to install. Setup alerts you if there is insufficient free space on the drive that contains your system folder.

7 On the Start Installation page, choose Next to start the installation.

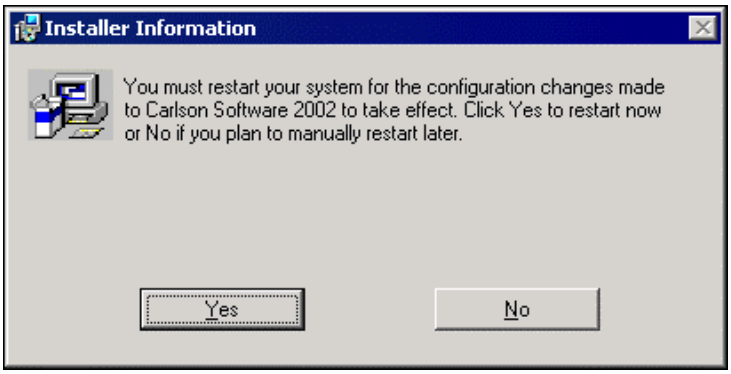


8 The Updating System dialog box is displayed while Agstar is installed.



9 When the installation is complete, the Setup Complete dialog box is displayed. Choose Finish to exit the installation program.

10It is strongly recommended that you restart your computer at this point in order for the new configuration settings to take effect.



Do one of the following:

Choose Yes to restart your computer now.

Choose No to manually restart your computer at another time.

If you do not restart your computer, you may have problems running Agstar.

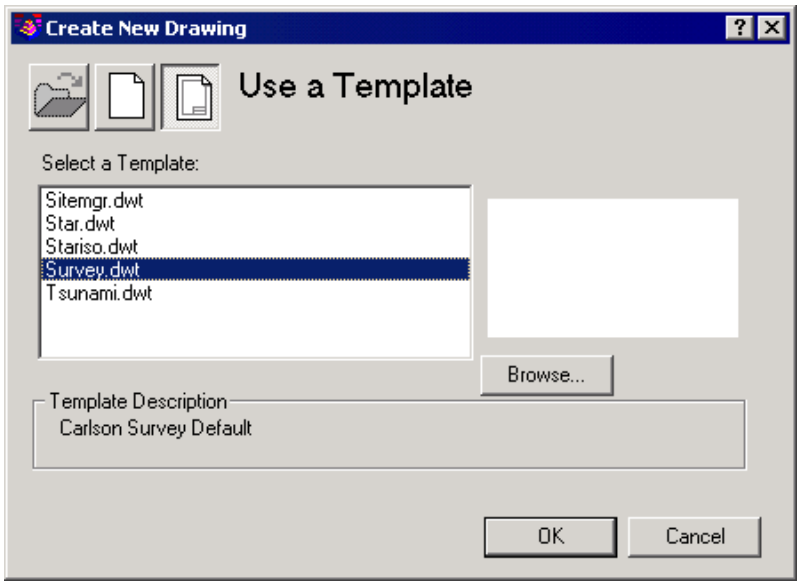
Congratulations! You have successfully installed Agstar. You are now ready to register your product and start using the program. To register the product, double-click the Agstar icon on your desktop and follow the instructions.

Command Entry

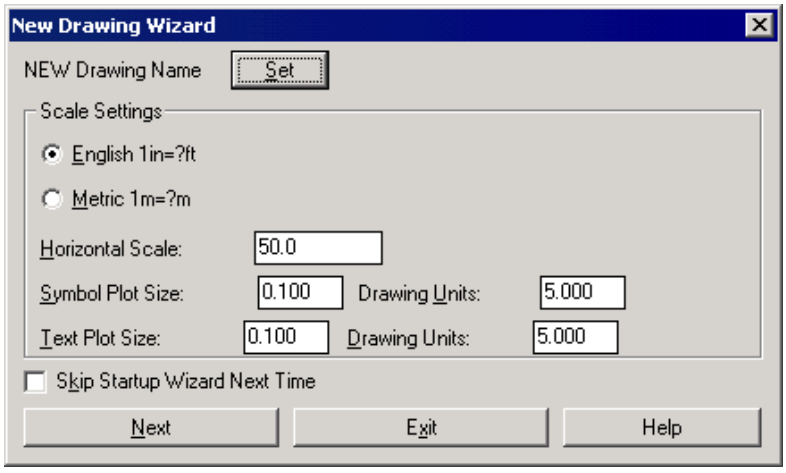
Commands may be issued by selecting an entry from a pull-down menu, clicking a toolbar button, or by typing a command at the command prompt. Pressing Enter at the command prompt repeats that last command. Pull-down menus have a row of header names across the top of the screen. Selecting one of these header names displays the possible commands under that name. The pull-down menus are the primary method for command selection. This manual is organized by the contents of each pull-down menu. Pull-down menus may sometimes be referred to as drop-down menus.

Startup Wizard

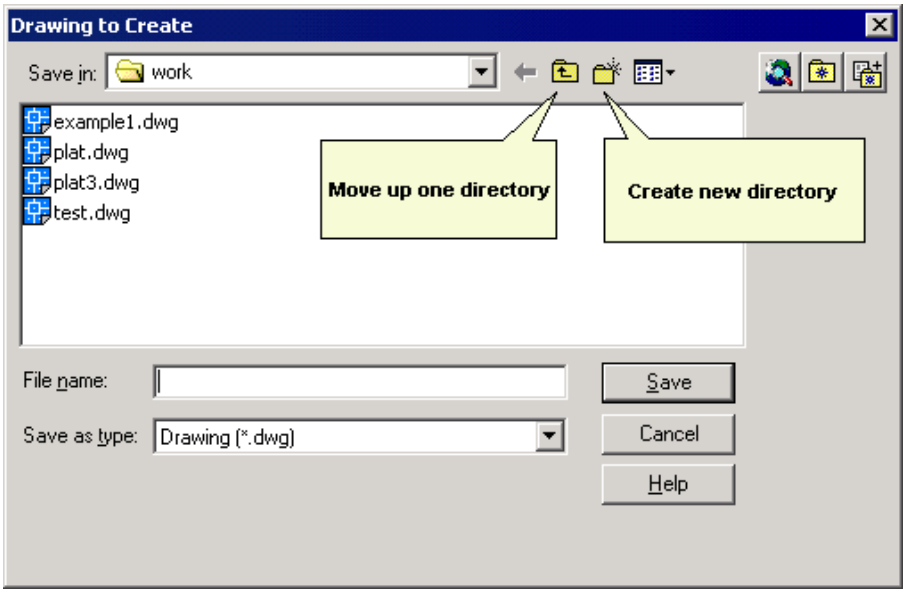
For creating a new drawing in Agstar, the Startup Wizard can guide you through starting and setting up the drawing. This wizard is optional and can be turned on or off in the Configure Survey — General Settings command. You can also exit out of the Startup Wizard at any time.



When the New drawing command is executed, you first get the standard AutoCAD choice of "Start from Scratch", "Use a Template" or "Use a Wizard". Typically, you want to the "Use a Template" option and choose the drawing template (SURVEY.DWT). The drawing template will set of some basic drawing parameters such as default layer names.



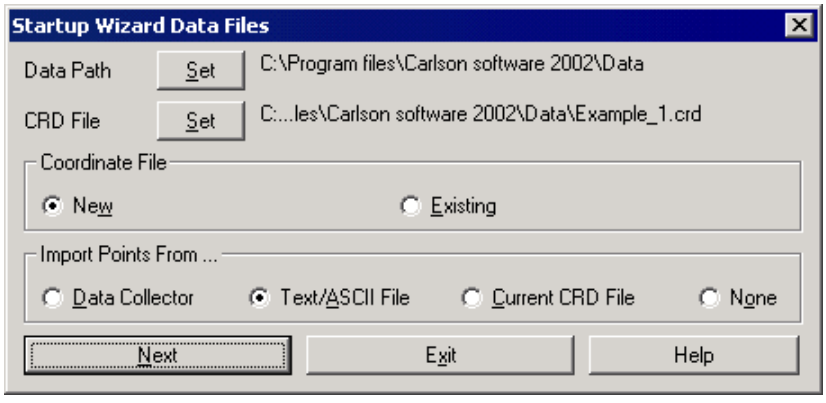
After selecting the AutoCAD new drawing option, the New Drawing Wizard dialog box opens. The Startup Wizard begins with a dialog to set the drawing name and scale. The first step to do is set the drawing (.dwg) name by picking the Set button. This brings up the file selection dialog. Change to the directory/folder ("Save in" field) where you want to store the drawing. You can either select an existing folder or create a new folder. To select an existing folder, pull down the Save in field to select a folder or drive, click the Move Up icon next to the Save in field and/or the pick the folder name from the list. To create a new folder, pick the Create New Folder icon to the right of the Save in field. Then type in the drawing name in the File name field and click the Save button.



After setting the drawing name, you can set the drawing horizontal scale, symbol size, text size and unit mode (English or Metric). Then click the Next button.

The next startup dialog sets the Data Path and CRD File. The Data Path is the folder where Agstar will store the data files such as raw (.RW5) files and profile (.PRO) files. The Set button for the Data Path allows you to select an existing folder or create a new folder. See the Set Data Directory command for more information. The coordinate (.CRD) File is the coordinate file for storing the point data. There is an option to create a new or existing coordinate (.CRD) file. The new option will erase any point data that is found in the specified CRD file. The existing option will retain any point data in the specified coordinate (.CRD) file. If the specified coordinate (.CRD) file does not exist, the wizard will create a new file.

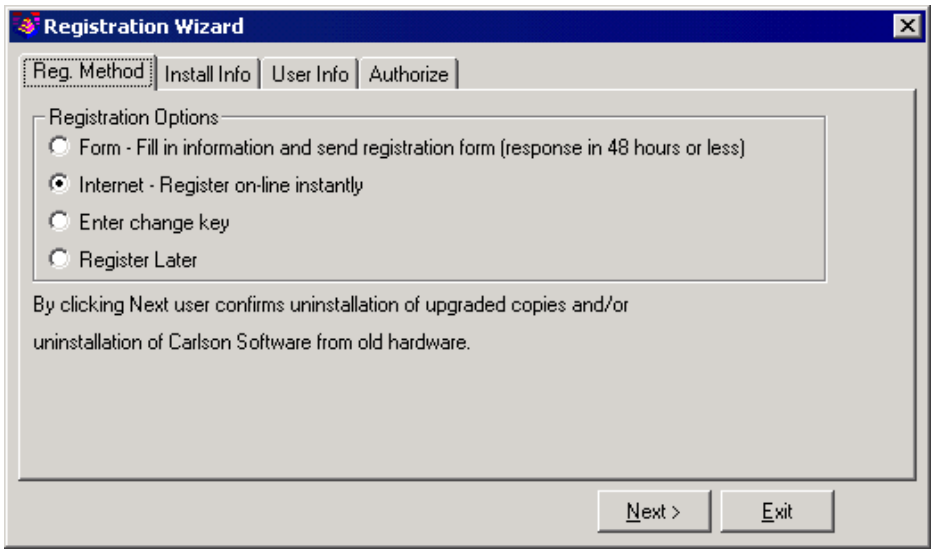
The next wizard step depends on the Import Points option. The Data Collector option will start the data collection routines to download data from a collector. The Text/ASCII option will import point data from a text/ASCII file. See the Data Collection and Import Text/ASCII File commands for more information on running these routines. If the None option is set, then the Startup Wizard is finished.



Once point data has been imported from the data collector or text/ASCII file, the wizard guides you through drawing the points. There are options to run Draw/Locate Points, Field To Finish or None. If None is selected, then the Startup Wizard is finished. Draw/Locate Points will import the points into the drawing using the same symbol and layer for all the points. From the Draw/Locate Points dialog, set the symbol, layer and point attributes to draw (description, elevation) and then pick the Draw All button. The Field To Finish command will import the points into the drawing using different layers and symbols depending on the point descriptions that refer to the code table defined in Field to Finish. Also Field to Finish can draw linework. See the Draw/Locate Point and Field To Finish commands for more information on running these routines. After drawing the points, the wizard will zoom the display around the points. Then the wizard is finished.

AuthorizingAgstar

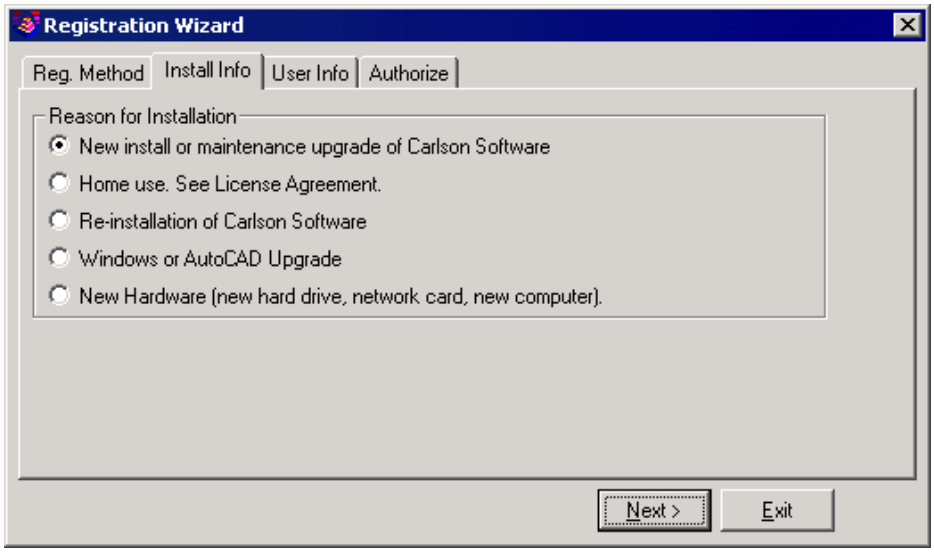
The first time you start Agstar, the Registration Wizard is displayed.



1 Carlson Software has installed an automated procedure for registering your software license. Change keys are no longer given over the telephone. Please choose one of the following registration methods.

- **Form**: This method allows you to fill out a form that you can print out and fax or mail to Carlson Software for registration.
- **Internet**: If your computer is online, you may register automatically over the Internet. Your information is sent to a Carlson Software server , validated and returned in just a few seconds. If you are using a dial-up connection, please establish this connection before attempting to register.
- **Enter change key**: Choose this method after you have received your change key from Carlson Software (if you previously used the Form method above).
- **Register Later**: Choose this method if your want to register later. You may run Agstar for 30 days before you are required to register.

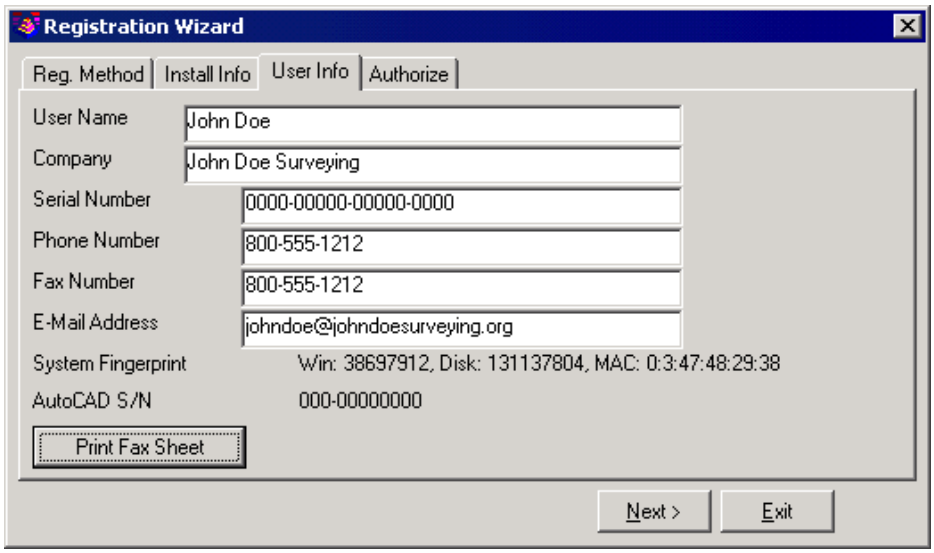
2 After you choose the registration method, press Next



3 Choose the reason for installation. The very first time you install Agstar is the only time you will choose the first reason. All subsequent installations require a choice from the remaining options.

- **New install or maintenance upgrade of Carlson Software**: If you are installing Agstar for the first time, choose this reason.
- **Home use. See License Agreement**: Choose this reason if you are installing on your home computer. See your license agreement for more details!
- **Re-Installation of Carlson Software**: Choose this reason if you are reinstalling on the same computer with no modifications.
- **Windows or AutoCAD upgrade**: Choose this reason if you have reinstalled Agstar after installing a new version of Microsoft Windows.
- **New Hardware**: Choose this reason if you are installing Agstar on a new computer or if your existing computer has had some of its hardware replaced such as the hard disk, network adapter, etc.

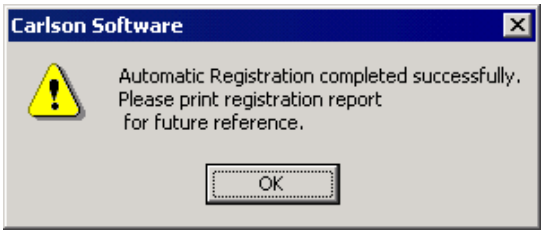
4 After you choose the reason for installation, press Next



5 Next, enter the required information into the dialog.

If you are using the Form method, press the Print Fax Form button to print out the form. You may fax this form to the number printed on the form or mail it to Carlson Software, 102 W. Second St., Suite 200 Maysville, KY 41056-1003.

If you are using the Internet method, press Next. After a few seconds, your registration will complete. If your registration is successful, you will receive a message such as the one below. If your registration is unsuccessful, please note the reason why and try again. Keep in mind that each serial number may be registered to a single computer only.



If you do not have access to the internet and do not have a printer, you must write down the information from the User Info tab (shown above) and fax it to 606-564-9525 or mail it to Carlson Software, 102 W. Second St., Suite 200 Maysville, KY 41056-1003.

License Agreement

Copyright 1992-2012 Carlson Software All Rights Reserved

CAUTION! READ THIS NOTICE BEFORE USING SOFTWARE

Please read the following Software License Agreement before using the SOFTWARE. Using this SOFTWARE indicates that you have accepted its terms and conditions.
Carlson AgStar 2013

END-USER LICENSE AGREEMENT FOR CARLSON SOFTWARE

IMPORTANT-READ CAREFULLY: This Carlson Software End-User License Agreement ("EULA") is a legal agreement between you (either an individual or a single entity) and Carlson Software, Inc for the software accompanying this EULA, which includes computer software and may include associated media, printed materials, and "online" or electronic documentation ("SOFTWARE PRODUCT" or "SOFTWARE"). By exercising your rights to use the SOFTWARE, you agree to be bound by the terms of this EULA. If you do not agree to the terms and conditions of this EULA, you may not use the SOFTWARE. IF YOU DO NOT AGREE TO THE TERMS AND CONDITIONS OF THIS EULA, DO NOT INSTALL OR USE ANY PART OF THE SOFTWARE.

Carlson Software, Inc., referred to as "LICENSOR", develops and/or licenses proprietary computer programs and sells use licenses for such proprietary computer programs together with or apart from accompanying copyrighted material and documentation and;

End User desires to obtain the benefits thereof and, in return for which, is willing to abide by the obligations and fee agreements applicable to LICENSOR's use licenses in LICENSOR's proprietary computer programs.

For good and valuable consideration, including but not limited to license grant in accordance with this Agreement by LICENSOR to End User's covenant regarding LICENSOR's proprietary rights, LICENSOR agrees to permit End User to utilize materials representing LICENSOR's product or products subject to the following terms and conditions:

1. License Grant: Subject to the terms, conditions and limitations of this EULA, LICENSOR hereby grants End User a personal, limited, non-exclusive, non-transferable, license to utilize the Software Product you have purchased. The license granted in this EULA creates no license, express or implied, to any other intellectual property of Licensor, except for the specific Software Product which they have lawfully purchased from LICENSOR.
This EULA grants you the following rights: You may install and use one copy of the SOFTWARE PRODUCT, or any prior version for the same operating system, on a single computer. The primary user of the computer on which the SOFTWARE PRODUCT is installed may make a second copy for his or her exclusive use on a portable computer.
Storage/Network Use. You may also store or install a copy of the SOFTWARE PRODUCT on a storage device, such as a network server, used only to install or run the SOFTWARE PRODUCT on your other computers over an internal network; however, you must acquire and dedicate a license for each separate computer on which the SOFTWARE PRODUCT is installed or run from the storage device. A license for the SOFTWARE PRODUCT may not be shared or used concurrently on different computers.

2. Exclusive Source. End User shall obtain all LICENSOR authorized product materials through LICENSOR or LICENSOR'S authorized representative and no other source. LICENSOR authorized product materials include, but are not limited to, manuals, license agreements and media upon which LICENSOR's proprietary computer programs are recorded. End User shall make no copies of any kind of any of the materials furnished by LICENSOR or LICENSOR's authorized representative, except as specifically authorized to do so in this EULA. End User is not entitled to make archival copies of those portions of LICENSOR's product(s) that are provided on a machine readable media.

3. Proprietary Rights of Licensor. End User agrees that LICENSOR retains exclusive ownership of the trademarks and service marks represented by its company name and logo and all of the documentation and computer recorded data related thereto. End User also agrees that all techniques, algorithms, and processes contained in LICENSOR's computer program products or any modification or extraction thereof constitute TRADE SECRETS OF LICENSOR and will be safeguarded by End User, but in no event shall End User exercise less than due diligence and care in accordance with the laws of the country of purchase and International Law, whichever operates to best protect the interests of LICENSOR. End User shall not copy, reproduce, re-manufacture or in any way duplicate all or any part of LICENSOR products WHETHER MODIFIED OR TRANSLATED INTO ANOTHER LANGUAGE OR NOT, or in any documentation, or in any other material provided by LICENSOR in association with LICENSOR's computer program products regardless of what manner of storage and retrieval the product exists, except as specified in this Agreement and in accordance with the terms and conditions of this Agreement which remain in force. End User agrees that in the event End User breaches this EULA, End User will be liable for damages as may be determined by a court of competent jurisdiction.

4. Restrictions. End User's rights and obligations under this EULA are nonexclusive and personal in nature, and the intellectual property Licensor grants to End User is subject to applicable law other than bankruptcy law. End User may not transfer or assign the SOFTWARE, rights under this EULA or accompanying user documentation, or any updates of the SOFTWARE which may be provided under this EULA, to a third party unless End User receives written consent from Licensor at least 30 days prior to the completion of transfer. Licensor reserves the right to deny transfer or assignment if, in its sole discretion, Licensor determines the transfer not to be a necessity. Whether or not a transfer or assignment is allowed shall be determined in Licensor's sole discretion after taking into consideration certain factors to find the existence of a necessity including, but not limited to, merger or acquisition of an entity, complete asset acquisition, change of control, severe economic hardship, severe loss of human resources or significant loss in business divisions, or winding down of entity affairs.

If Carlson consents to a transfer, such transfer shall be allowed only as a one-time permanent transfer of this EULA and Software to another end user, provided the initial End User retains no copies or previous versions of the Software. The transfer must include all of the Software, including all component parts, any media and printed materials, any upgrades, this EULA, and any associated license key. The transfer may not be an indirect transfer, such as a consignment, rental or lease. No corresponding Maintenance Agreement rights shall transfer with the SOFTWARE transfer to the subsequent end user. Prior to the transfer, the subsequent end user receiving the Software from the initial End User must agree to all terms of this EULA, with the added condition that no further transfers to third parties are permitted for any reason whatsoever, and shall agree to the terms and conditions of a new Maintenance Agreement with Licensor.

You may not reverse engineer, decompile, or disassemble the SOFTWARE or alter the images utilized in the SOFTWARE and user documentation. The SOFTWARE PRODUCT is licensed as a single product. Its component parts may not be separated for use on more than one computer. You shall communicate to any individual user in your facility that they are bound by the restrictions of this license agreement may not copy or alter the SOFTWARE for use outside End User's facilities.
Upgrades. If you purchase an upgrade of a SOFTWARE PRODUCT and you use it on different machine from one where upgraded SOFTWARE PRODUCT was used, use of original SOFTWARE PRODUCT must be discontinued and confirmed within 30 days. If such use is not discontinued, it is a material breach of this EULA and LICENSOR shall be entitled to all remedies available to it under this EULA, and under the laws of Kentucky, USA.

5. Security Mechanisms. Licensor and its affiliated companies take all legal steps to eliminate piracy of their software products. In this context, the Software Product may include a security mechanism that can detect the installation or use of illegal copies of the Software Product, and collect and transmit data about those illegal copies. Data collected will not include any customer data created with the Software. By using the Software Product, you consent to such detection and collection of data, as well as its transmission and use if an illegal copy is detected. Licensor also reserves the right to use a hardware lock device, license administration software, and/or a license authorization key to control access to the Software. You may not take any steps to avoid or defeat the purpose of any such measures. Use of any Software without any required lock device or authorization key provided by Licensor is

prohibited.

6. Audit Rights. End User agrees that LICENSOR has the right to require an audit (electronic or otherwise) of the LICENSOR Materials and the Installation thereof and access thereto. As part of any such audit, LICENSOR or its authorized representative will have the right, on fifteen (15) days' prior notice to End User, to inspect End User's records, systems and facilities, including machine IDs, serial numbers and related information, to verify that the use of any and all LICENSOR Materials is in conformance with this Agreement. End User will provide full cooperation to enable any such audit. If LICENSOR determines that End User's use is not in conformity with this EULA, End User will obtain immediately and pay for a valid license to bring End User's use into compliance with this EULA and other applicable terms and pay the reasonable costs of the audit. In addition to such payment rights, LICENSOR reserves the right to seek any other remedies available at law or in equity, whether under this Agreement or otherwise.

7. Warranty. THE PRODUCT IS PROVIDED "AS IS" WITH ALL FAULTS. TO THE MAXIMUM EXTENT PERMITTED BY LAW, LICENSOR HEREBY DISCLAIMS ALL WARRANTIES, WHETHER EXPRESS OR IMPLIED, INCLUDING WITHOUT LIMITATION IMPLIED WARRANTIES OF MERCHANITIBILITY, FITNESS FOR A PARTICULAR PURPOSE, AND WARRANTIES THAT THE PRODUCT IS FREE OF DEFECTS AND NON-INFRINGING, WITH REGARD TO THE SOFTWARE, AND THE ACCOMPANYING WRITTEN MATERIALS. YOU BEAR ENTIRE RISK AS TO SELECTING THE PRODUCT FOR YOUR PURPOSES AND AS TO THE QUALITY AND PERFORMANCE OF THE PRODUCT. THIS LIMITATION WILL APPLY NOTWITHSTANDING THE FAILURE OF ESSENTIAL PURPOSE OF ANY REMEDY. In any event, LICENSOR will not honor any warranty shown to exist for hich inaccurate or incorrect identifying data has been provided to LICENSOR. The product(s) provided are intended for commercial use only and should not be utilized as the sole data source in clinical decisions as to levels of care.

8. LIMITATION OF LIABILITY. EXCEPT AS REQUIRED BY LAW, LICENSOR AND ITS DISTRIBUTORS, DIRECTORS, LICENSORS, CONTRIBUTORS AND AGENTS (COLLECTIVELY, THE "LICENSOR GROUP") WILL NOT BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, CONSEQUENTIAL OR EXEMPLARY DAMAGES ARISING OUT OF OR IN ANY WAY RELATING TO THIS EULA OR THE USE OF OR INABILITY TO USE THE PRODUCT, INCLUDING WITHOUT LIMITATION DAMAGES FOR LOSS OF GOODWILL, WORK STOPPAGE, LOST PROFITS, LOSS OF DATA, AND COMPUTER FAILURE OR MALFUNCTION, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGES AND REGARDLESS OF THE THEORY (CONTRACT, TORT OR OTHERWISE) UPON WHICH SUCH CLAIM IS BASED. THE LICENSOR GROUP'S COLLECTIVE LIABILITY UNDER THIS AGREEMENT WILL NOT EXCEED THE GREATER OF $500 (FIVE HUNDRED DOLLARS) AND THE FEES PAID BY YOU UNDER THIS LICENSE (IF ANY).

9. Update Policy. LICENSOR may, from time to time, revise the performance of its product(s) and in doing so, incur NO obligation to furnish such revisions to any End User nor shall it warrant or guarantee that any revision to the SOFTWARE will perform as expected by the End User on End User's equipment. At LICENSOR's option, LICENSOR may provide such revisions to the End User.

10. Customer Service. Although it is the LICENSOR's customary practice to provide reasonable assistance and support in the use of its products to its customers, LICENSOR shall not be obligated to any End User to provide technical assistance or support through this Agreement and may at LICENSOR's sole election charge a fee for customer support.

11. Termination of End User License. If any one or more of the provisions of this Agreement is breached, the license granted by this Agreement is hereby terminated. In the event of such termination, all rights of the LICENSOR shall remain in force and effect. Any protected health information data of End User maintained on LICENSOR'S data base shall upon reasonable notice to End User and at the discretion of LICENSOR may be destroyed.

12. Copyright. The SOFTWARE (including, but not limited to, any images, photographs, animations, video, audio, music and or text incorporated into the SOFTWARE), and all intellectual property rights associated with it, whether exists in a tangible media or in an electronic image media is owned by LICENSOR and is protected by United States copyright laws and international treaty provisions and all other commonwealth or national laws. LICENSOR reserves all intellectual property rights in the Products, except for the rights expressly granted in this Agreement. You may not remove or alter any trademark, logo, copyright or other proprietary notice in or on the Product. This license does not grant you any right to use the trademarks, service marks or logos of LICENSOR or its licensors. You may not copy any user documentation accompanying the SOFTWARE.

13. Injunctive Relief. It is understood and agreed that, notwithstanding any other provision of this Agreement, LICENSOR has the unequivocal right to obtain timely injunctive relief to protect the proprietary rights of LICENSOR.

14. Entire Agreement. This EULA constitutes the entire agreement between the parties and supersedes any prior agreements. This EULA may only be changed by mutual written consent.

15. End User Agreement Acknowledgment. The End User hereby accepts all the terms and conditions of this Agreement without exception, deletion, alteration. End User acknowledges they are authorized to enter this agreement on behalf of any organization for which the license is sought. Any unauthorized use of LICENSOR products will be considered a breach of this Agreement, subject to liquidated damages and otherwise unlawful and willful infringement of LICENSOR's trade secrets and/or proprietary products.

16. Payment and Refund Policy. The use of the SOFTWARE herein is deemed a commercial use and under the terms of this license agreement End User shall not be entitled to any refund of purchase price. End User agrees to pay all user fees promptly. LICENSOR is authorized by End User to suspend any further access to SOFTWARE in the event fees are not fully paid. End user entity shall promptly pay any and all access and use charges incurred regardless of the end user. End user is responsible for protecting any pass word and user identity supplied to End User.

17. Loss/Theft/Misuse. End user shall promptly report to LICENSOR the theft or other loss of any password and/or user identity required to access SOFTWARE. LICENSOR shall not be responsible for maintaining the integrity of End User data in the event that end user's data base is accessed and/or altered by an unauthorized end user due to the failure of licensed End User to protect its password or user identity. End User shall be responsible for any costs incurred by LICENSOR due to the negligence or reckless disregard of End User's failure to protect its password or user identity.

18. Civil/Criminal Investigation. End user shall fully cooperate with LICENSOR and or any person authorized by LICENSOR (including local, state, or federal law enforcement officials) to investigate any alleged theft, misuse or unauthorized use of SOFTWARE or data related thereto.

19. U.S. Government Restricted Rights. The SOFTWARE PRODUCT and documentation are provided with RESTRICTED RIGHTS. Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (b)(1)(ii) and (c) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs (c)(1) and (2) of the Commercial Computer Software-Restricted Rights at 48 CFR 52.227-19, as applicable.

20. Governing Law. This EULA shall be governed and construed in accordance with the laws of the Commonwealth of Kentucky, USA.

Technical Support

Discussion Groups

Carlson Software operates user discussion groups. The NNTP address is news.carlsonsw.com. Visit our website for complete details on how to connect to these discussion groups.

Electronic Mail

The technical support email address is support@carlsonsw.com

Internet

The Knowledge base is available at update.carlsonsw.com/kbase

Program updates and patches are available at update.carlsonsw.com

Technical support documents are available at www.carlsonsw.com

Phone or Facsimile

Phone: 606-564-5028

Fax: 606-564-6422

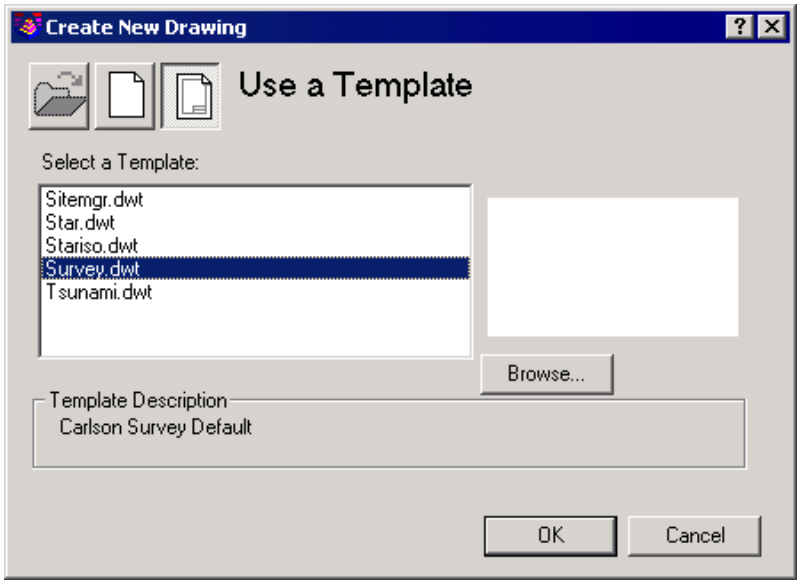
Fax for registrations only: 606-564-9525

Please submit your company name, product version, and serial number with all support inquiries

File Commands 2

New

This command allows you to create a new drawing file.



This command defines the settings for a new drawing. There are two methods that you can use to create a new drawing. (The first option, Open a Drawing, is not available from the NEW command. To open an existing drawing, use the OPEN command.) Choose one of the icons at the top of the dialog box.

1 Under Start from Scratch, you can start a new drawing file.

This command starts a new drawing using default settings defined in either the surv.dwt or surviso.dwt template, depending on the measurement system you've chosen. You cannot modify the surv.dwt or surviso.dwt templates. To start a new drawing based on a customized template, see Use a Template.

- **English**: This option starts a new drawing based on the Imperial measurement system. The drawing is based on the surv.dwt template, and the default drawing boundary (the drawing limits) is 12 × 9 inches.
- **Metric**: This option starts a new drawing based on the metric measurement system. The drawing is based on the surviso.dwt template, and the default drawing boundary (the drawing limits) is 429 × 297 millimeters.

2 Under Use a Template, you can start a new drawing based on a customized template.

This command creates a new drawing using the settings defined in a template drawing you select. Template drawings store all the settings for a drawing and may also include predefined layers, dimension styles, and views. Template drawings are distinguished from other drawing files by the .dwt file extension. They are normally kept in the template directory. Several template drawings are included with AgStar. You can make additional template drawings by changing the extensions of drawing file names to .dwt.

- **Select a Template**: This option lists all template files that currently exist in the drawing template file location, which is specified in the Options dialog box. Choose a file to use as a starting point for your new drawing. A preview image of the selected file is displayed to the right.
- **Browse**: This option displays the Select a Template File dialog box (a standard file selection dialog box) where you can access template files in other directions.

Menu Location: File
Prerequisite: None
Keyboard Command: NEW

Open

This command allows you to open an existing drawing file. AgStar displays the Select File dialog box (a standard file selection dialog box). Select a file and click Open.

Menu Location: File
Prerequisite: None
Keyboard Command: OPEN

Close

Function

This command allows you to close the current drawing. AgStar closes the current drawing if there have been no changes since the drawing was last saved. If you have modified the drawing, the program prompts you to save or discard the changes. You can close a file that has been opened in Read-only mode if you have made no changes or if you are willing to discard changes. To save changes to a read-only file, you must use the SAVEAS command.

Menu Location: File

Prerequisite: None

Keyboard Command: CLOSE

Save

If the drawing is named, Carlson Survey saves the drawing without requesting a file name. If the drawing is unnamed, the program displays the Save Drawing As dialog box (see SAVEAS) and saves the drawing with the file name you specify. If the drawing is read-only, use the SAVEAS command to save the changed file under a different name. This command allows you to save the drawing under the current file name or a specified name

Menu Location: File
Prerequisite: None
Keyboard Command: SAVE or QSAVE

Save As

This command allows you to save an unnamed drawing with a file name or renames the current drawing.

AgStar displays the Save Drawing As standard file selection dialog box. Enter a file name and type. You can select any of the following file types:

- AutoCAD 2000 (∗.dwg)
- AutoCAD R14/LT 98/LT 97 Drawing (∗.dwg)
- AutoCAD R13/LT 95 Drawing (∗.dwg)
- Drawing Template File (∗.dwt)
- Carlson Software 2002 DXF (∗.dxf)
- AutoCAD R14/LT 98/LT 97 DXF (∗.dxf)
- AutoCAD R13/LT 95 DXF (∗.dxf)
- AutoCAD R12/LT2 DXF (∗.dxf)

AgStar saves the file under the specified file name. If the drawing is already named, the program saves the drawing to the new file name. If you save the file as a drawing template, the program displays the Template Description dialog box, where you can provide a description for the template and set the units of measurement.

Saving a drawing in Release 14/LT 98/LT 97 format is subject to the following limitations:

- Hyperlinks are converted to Release 14 attached URLs.
- Database links and freestanding labels are converted to Release 14 links and displayable attributes.
- Database attached labels are converted to MText and leader objects, and their link information is not available. Attached labels are restored if you open the drawing in AutoCAD 2000 or later.
- Lineweight information is not available. Lineweights are restored if you open the drawing in AutoCAD 2000 or later.

Saving a drawing in Release 13/LT 95 format is subject to the following limitations:

- Lightweight polyline and hatch patterns are converted to R13 polylines and hatch patterns.
- Raster objects are displayed as bounding boxes. Raster objects are restored if the drawing is opened in AutoCAD 2000 or later.
- Draw order information is not applied for display or print.
- Xrefs that have been clipped with a boundary box are displayed in full as attached xrefs because Release 13 does not support xref clipping. Clipping is restored if the drawing is opened in AutoCAD 2000 or later.

Saving a drawing in Release 12/LT 2 DXF format is subject to the following limitations:

- Lightweight polylines and hatch patterns are converted to R12 polylines and hatch patterns.
- All solids, bodies, regions, ellipses, leaders, multilines, rays, tolerances, and xlines are converted to lines, arcs, and circles as appropriate.
- Groups, complex linetypes, OLE objects, and preview images are not displayed.
- Many objects are lost if you save a drawing as Release 12 and open it later in AutoCAD 2000 or later.

Menu Location: File
Prerequisite: None
Keyboard Command: SAVEAS

Plot

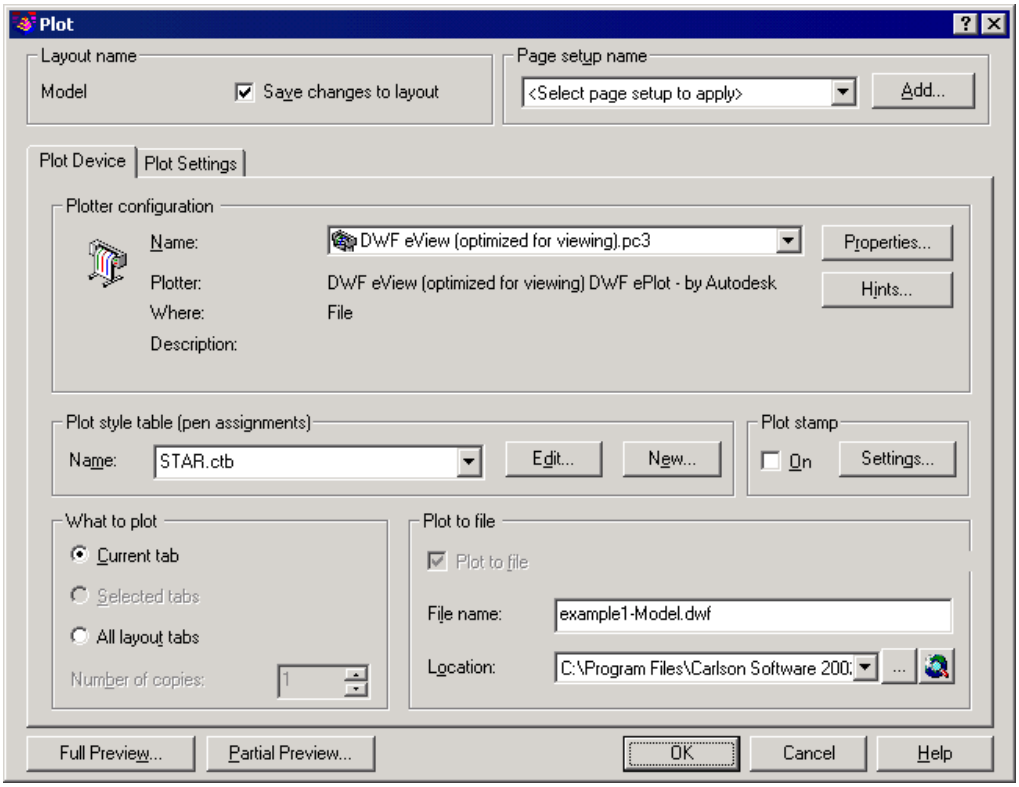
This command allows you to plot a drawing to a plotting device or file.

AgStar displays the Plot dialog box. Choose OK to begin plotting with the current settings and display the Plot Progress dialog box.

1 The Plot dialog box includes the tabs, Plot Device and Plot Settings, and several options to customize the plot.

- **Layout Name**: This option displays the current layout name or displays "Selected layouts" if multiple tabs are selected. If the Model tab is current when you choose Plot, the Layout Name shows "Model."
- **Save Changes to Layout**: This option saves the changes you make in the Plot dialog box in the layout. This option is unavailable if multiple layouts are selected.
- **Page Setup Name**: This option displays a list of any named and saved page setups. You can choose to base the current page setup on a named page setup, or you can add a new named page setup by choosing Add.
- **Add**: This option displays the User Defined Page Setups dialog box. You can create, delete, or rename named page setups.

2 Under the Plot Device Tab you can specify the plotter to use, a plot style table, the layout or layouts to plot, and information about plotting to a file.



- **Plotter Configuration**: This field displays the currently configured plotting device, the port to which it's connected or its network location, and any additional user-defined comments about the plotter. A list of the available system printers and PC3 file names is displayed in the Name list. An icon is displayed in front of the plotting device name to identify it as a PC3 file name or a system printer.
- **Properties**: The option displays the Plotter Configuration Editor (PC3 Editor), where you can modify or view the current plotter configuration, ports, device, and media settings.
- **Hints**: This option displays information about the specific plotting device.
- **Plot Style Table (Pen Assignments)**: This option sets the plot style table, edits the plot style table, or creates a new plot style table.
- **Name**: This option displays the plot style table assigned to the current Model tab or layout tab and a list of the currently available plot style tables. If more than one layout tab is selected and the selected layout tabs have different plot style tables assigned, the list displays "Varies."
- **Edit**: This option displays the Plot Style Table Editor, where you can edit the selected plot style table.
- **New**: This option displays the Add-a-Plot-Style-Table wizard, which you can use to create a new plot style table.
- **Plot Stamp**: This option places a plot stamp on a specified corner of each drawing and/or logs it to a file.
- **On**: This options turns on plot stamping.
- **Settings**: This option displays the Plot Stamp dialog box, where you can specify the information you want applied to the plot stamp, such as drawing name, date and time, and plot scale.
- **What to Plot**: This field defines the tabs to be plotted.
- **Current Tab**: This option plots the current Model or layout tab. If multiple tabs are selected, the tab that shows its viewing area is plotted.
- **Selected Tabs**: This option plots multiple preselected Model or layout tabs. To select multiple tabs, hold down CTRL while selecting the tabs. If only one tab is selected, this option is unavailable.
- **All Layout Tabs**: This option plots all layout tabs, regardless of which tab is selected.

• **Number of Copies**: This option denotes the number of copies that are plotted. If multiple layouts and copies are selected, any layouts that are set to plot to a file or AutoSpool produce a single plot.

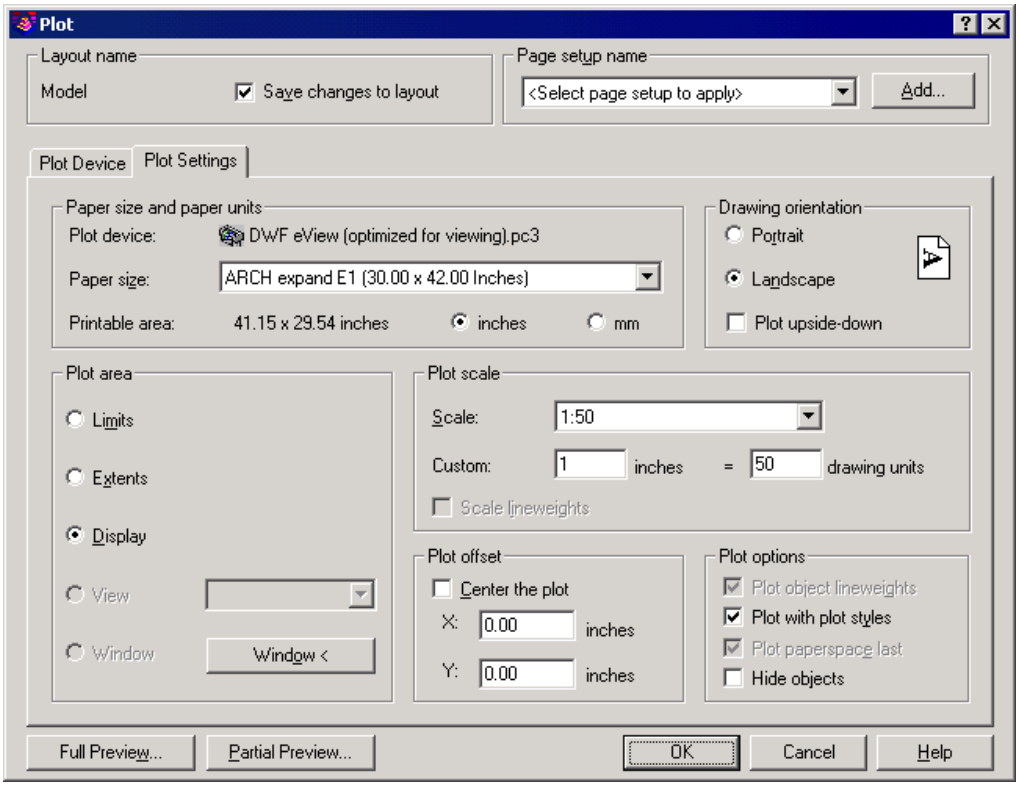
• **Plot to File**: This option plots output to a file rather than to the plotter.

• **File Name**: This option specifies the plot file name. The default plot file name is the drawing name and the tab name, separated by a hyphen, with a .plt file extension.

• **Location**: This option displays the directory location where the plot file is stored. The default location is the directory where the drawing file resides.

• **[...]**: This option displays a standard Browse for Folder dialog box, where you can choose the directory location to store a plot file.

3 Under the Plot Settings Tab you specify paper size, orientation, plot area and scale, offset, and other options.



• **Paper Size and Paper Units**: This field displays standard paper sizes available for the selected plotting device. Actual paper sizes are indicated by the width (X axis direction) and height (Y axis direction). If no plotter is selected, the full standard paper size list is displayed and available for selection. A default paper size is set for the plotting device when you create a PC3 file with the Add-a-Plotter wizard. The paper size you select is saved with a layout and overrides the PC3 file settings. If you are plotting a raster image, such as a BMP or TIFF file, the size of the plot is specified in pixels, not in inches or millimeters.

• **Plot Device**: This field displays the name of the currently selected plot device.

• **Paper Size**: This field displays a list of the available paper sizes.

• **Printable Area**: This field displays the actual area on the paper that is used for the plot based on the current paper size.

• **Inches**: This option allows you to specify inches for the plotting units.

• **MM**: This option allows you to specify millimeters for the plotting units.

• **Drawing Orientation**: This option specifies the orientation of the drawing on the paper for plotters that support landscape or portrait orientation. You can change the drawing orientation to achieve a 0-, 90-, 180-, or 270-degree

plot rotation by selecting Portrait, Landscape, or Plot Upside-Down. The paper icon represents the media orientation of the selected paper. The letter icon represents the orientation of the drawing on the page.

- **Portrait**: This option orients and plots the drawing so that the short edge of the paper represents the top of the page.
- **Landscape**: This option orients and plots the drawing so that the long edge of the paper represents the top of the page.
- **Plot Upside-Down**: This option orients and plots the drawing upside down.
- **Plot Area**: This option specifies the portion of the drawing to be plotted.
- **Layout**: This option plots everything within the margins of the specified paper size, with the origin calculated from 0,0 in the layout. Available only when a layout is selected. If you choose to turn off the paper image and layout background on the Display tab of the Options dialog box, the Layouts selection becomes Limits.
- **Limits**: This option plots the entire drawing area defined by the drawing limits. If the current viewport does not display a plan view, this option has the same effect as the Extents option. Available only when the Model tab is selected.
- **Extents**: This option plots the portion of the current space of the drawing that contains objects. All geometry in the current space is plotted. AgStar may regenerate the drawing to recalculate the extents before plotting.
- **Display**: This option plots the view in the current viewport in the selected Model tab or the current paper space view in the layout.
- **View**: This option plots a previously saved view. You can select a named view from the list provided. If there are no saved views in the drawing, this option is unavailable.
- **Window**: This option plots any portion of the drawing you specify. If you select Window, the Window button becomes available. Choose the Window button to use the pointing device to specify the two corners of the area to be plotted or enter coordinate values.
- **Plot Scale**: This option controls the plot area. The default scale setting is 1:1 when plotting a layout. The default setting is Scaled to Fit when plotting a Model tab. When you select a standard scale, the scale is displayed in Custom.
- **Scale**: This option defines the exact scale for the plot. The four most recently used standard scales are displayed at the top of the list.
- **Custom**: This option creates a custom scale. You can create a custom scale by entering the number of inches or millimeters equal to the number of drawing units.
- **Scale Lineweights**: This option scales lineweights in proportion to the plot scale. Lineweights normally specify the linewidth of printed objects and are plotted with the linewidth size regardless of the plot scale.
- **Plot Offset**: This field specifies an offset of the plotting area from the lower-left corner of the paper. In a layout, the lower-left corner of a specified plot area is positioned at the lower-left margin of the paper. You can offset the origin by entering a positive or negative value. The plotter unit values are in inches or millimeters on the paper.
- **Center the Plot**: This option automatically calculates the X and Y offset values to center the plot on the paper.
- **X**: This field specifies the plot origin in the X direction.
- **Y**: This field specifies the plot origin in the Y direction.
- **Plot Options**: This field specifies options for lineweights, plot styles, and the current plot style table. You can select whether lineweights are plotted. By selecting Plot with Plot Styles, you plot using the object plot styles that are assigned to the geometry, as defined by the plot style table.
- **Plot object lineweights**: This option plots lineweights.
- **Plot with Plot Styles**: This option plots using the plot styles applied to objects and defined in the plot style table. All style definitions with different property characteristics are stored in the plot style tables and can be easily attached to the geometry. This setting can replace pen mapping in earlier versions of AutoCAD.

- **Plot Paperspace Last**: This option plots model space geometry first. Paper space geometry is usually plotted before model space geometry.

- **Hide Objects**: This option plots layouts with hidden lines removed for objects in the layout environment (paper space). Hidden line removal for model space objects in viewports is controlled by the Viewports Hide property in the Object Property Manager. This is displayed in the plot preview, but not in the layout.

- **Full Preview**: This option displays the drawing as it will appear when plotted on paper. To exit the print preview, right-click and choose Exit.

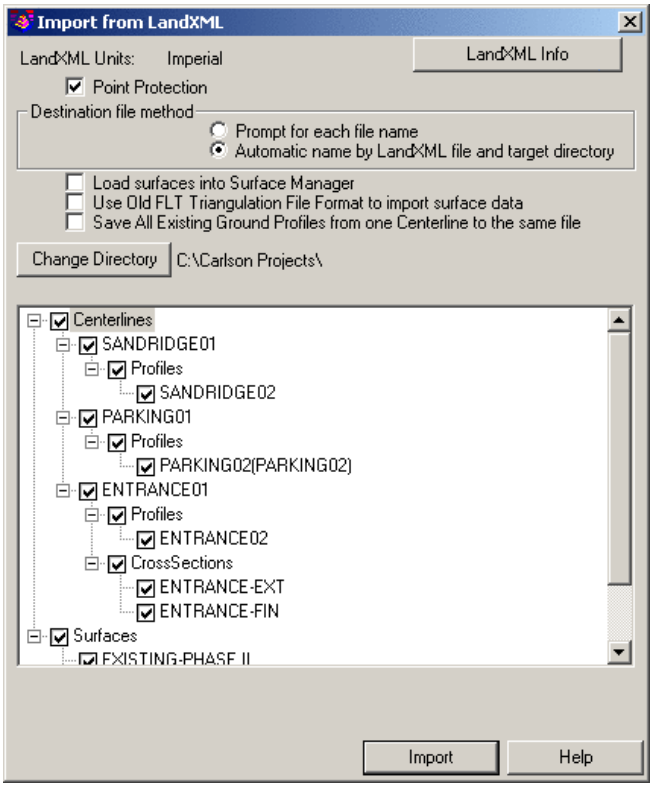
- **Partial Preview**: This option quickly shows an accurate representation of the effective plot area relative to the paper size and printable area. Partial preview also gives advance notice of any warnings that you might encounter when plotting. The final location of the plot depends on the plotter. Changes that modify the effective plot area include those made to the plot origin, which you define under Plot Offset on the Plot Settings tab. If you offset the origin so much that the effective area extends outside the preview area, the program displays a warning.

Menu Location: File
Prerequisite: None
Keyboard Command: PLOT

Import LandXML File

The Import LandXML File routine provides a mechanism where land-based data from other software applications (including Carlson Software) can be brought into a project and used for analysis and/or design purposes. To import a LandXML file, a series of dialog boxes are presented:

Select LandXML File: Specify the name of a LandXML file you wish to import.



LandXML Units: Indicates the Units of Measure associated with the incoming LandXML file (see the Unit Differences item below).

Point Protection: When enabled, you are prompted for a course of action if an existing LandXML file you've selected contains COGO points that have the same number(s) as those that already exist in the drawing. When disabled, existing point data in the project is updated with the values from the LandXML file.

Destination File Method: This option allows you to indicate how the incoming data file(s) are named as they are imported.

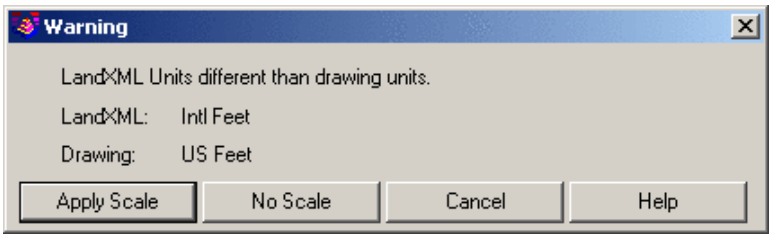
Load Surfaces into Surface Manager: When enabled, this option will automatically add surface model (TIN) data into the Surface Manager and graphically represents (draws) the surface model/contours according to the current settings found in the Triangulate & Contour command.

Use Old FLT Triangulation File Format to Import Surface Data: When enabled, the older ASCII-based Carlson ∗.FLT file format will be used in place of the newer and more efficient ∗.TIN file format.

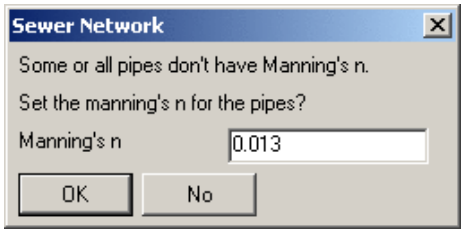
Save All Existing Ground Profiles from One Centerline to the Same File: When enabled, collections of existing ground profiles associated with a particular centerline are combined into a single ∗.PRO file.

Change Directory: This option allows you to adjust the folder location where the new data files will be written.

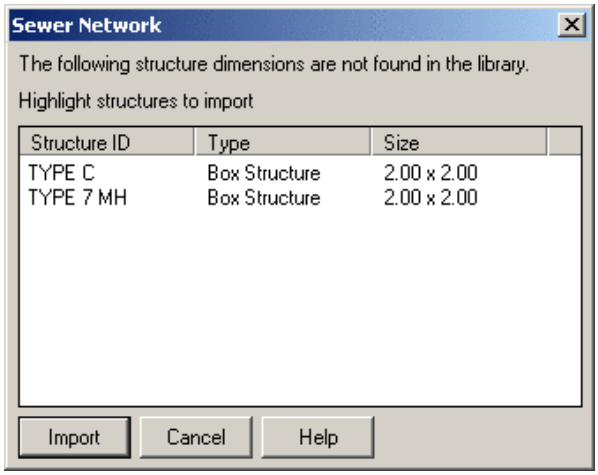
Import from LandXML: Enable or disable various entries that should used to produce the data files found within the LandXML file.



Unit Differences: If the Units of Measure specified in the LandXML file are different than those found in Drawing Setup, you will be prompted for a course of action.



Manning's "n": If you are importing sewer data from a LandXML file and if the LandXML file does not carry Manning's "n" values, you will be prompted to specify a default Manning's "n" value for all incoming sewer entities that don't already have a Mannings "n" value.



Import Structures: If you are importing sewer data from a LandXML file and structure values specified in the LandXML file do not exist in the Structure Library, you will be prompted to indicate the structure(s) that should be imported into the Structure Library. Use standard Windows **click**, **shift+click** and/or **ctrl+click** functionality to

select multiple structures at the same time.

Skip Invisible Triangles: This option applies to importing TIN surfaces from Civil 3D. When this option is active, triangles marked by Civil 3D as invisible or excluded are not imported.

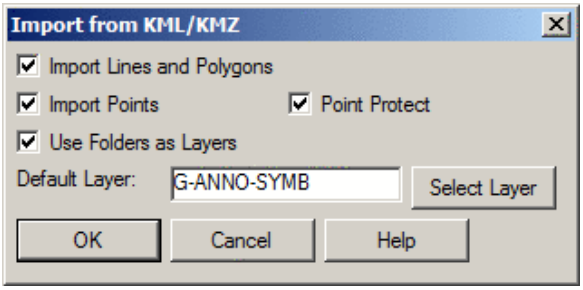
Note:

- The LandXML initiative is being driven by the land development industry as an acceptable means to share and transfer land data rather than the traditional graphical representation of that data. It also provides an effective means for transferring a variety data (points, centerlines, profiles, surface models, sewer data, *etc*). Another advantage of LandXML is that the LandXML data structure is CAD and software vendor neutral (meaning you don't have to own or use the CAD or software product used by your data provider).
- Visit http://www.landxml.org for additional information on the uses and acceptance of the LandXML initiative.

Pulldown Menu Location(s): File > Import
Keyboard Command: landxml_import
Prerequisite: A LandXML file to import

Import Google Earth File

The Import Google Earth File command allows you to insert a KML (Keyhole Markup Language or alternatively a KMZ) file of points (KML *Placemark*), polylines (KML *Path*) and closed polylines (KML *Polygon*) into your drawing. Throughout this discussion, *KML* will be used to also describe KMZ files unless explicitly noted.



Import Lines and Polygons: When this option is selected, KML Path and Polygon entries will be placed into the drawing as open or closed polylines, respectively.

Import Points: When this option is selected, KML Placemark entries will be placed into the drawing and active coordinate file.

Point Protect: When enabled, existing points in the active coordinate file will not be over-written.

Use Folders as Layers: When enabled, KML Folder entries will be used to create layer names in CAD and the supported KML options described above will be placed onto the layer that conforms the the Folder to which they belong.

Default Layer: The supported KML options described above that are not contained in a KML folder will be placed into the specified layer.

Note:

- Placemarks, paths or polygon entries that have an altitude value specified will be imported at the proper "Z" elevation in the CAD drawing.
- KML or KMZ files can be specified for the import process.

Prompts

Google Earth File to Read: *Select a previously saved KML or KMZ file.*

- To import a Google Earth image into your drawing, use the Place Google Earth Image command.
- To import a Google Earth terrain data into a Carlson TIN (surface model), use the Place Google Earth Image command.
- To export content from your drawing to a KML file, use the Export Google Earth File command.

Pulldown Menu Location: File > Import
Keyboard Command: kmlread
Prerequisite: A KML or KMZ file with Placemark, Path and/or Polygon information, an active coordinate file with an established projection zone through Drawing Setup.

Import/Export Topcon TIN File

The Export Topcon TIN File command writes a Topcon TIN file (.TN3) from a Carlson triangulation file (.TIN, .FLT). The routine first prompts for the Carlson file and then the Topcon file.

The Import Topcon TIN File command creates a Carlson Tin file (.TIN, .FLT) from a Topcon triangulation file (.TN3). The routine first prompts for the Topcon file and then the Carlson file.

The units (Feet or Meters) for the triangulation file are the current units set in Drawing Setup.

Pulldown Menu Location: File > Import/Export
Keyboard Command: topcon_tin, tn3_to_tin
Prerequisite: A triangulation file

Import/Export Trimble TTM File

These commands convert between Trimble TTM format triangulation files and Carlson format. First you select the source file to read and then the destination file to write.

Pulldown Menu Location: File->Import/Export
Keyboard Command: ttm2tin, tin2ttm
Prerequisite: File to convert

Write Polyline File

This command creates a polyline file that contains the point data of the select polylines. The objects supported by this tool include polylines, arcs and lines. Several different output formats are supported.

The Carlson format (.PLN) is a text file format that is used by some Carlson commands and by machine control (Carlson Grade, Dozer 2000, GradeStar) for the plan view. Each polyline begins with a line of "POLYLINE, Color number, *etc*". Then the points for the polyline are listed on separate lines in X,Y,Z format.

The DTM and Idan formats create linework files for the DTM and Idan programs.

The MicroStation format (.txt) can be imported into MicroStation. This format has the coordinates as space delimited for each polyline point. There is an extra column with a 1 or 0 where 1 specifies the start of a new polyline.

The Moss format creates a INP file for the MX/MOSS Genio program.

The Peabody format is a company specific format for Peabody Energy.

The Topcon format creates a Topcon LN3 file.

The 12D format creates a file format compatible with the 12d modeling program.

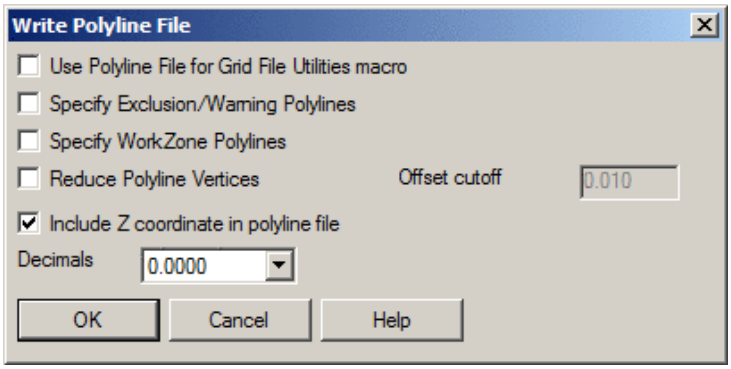
Note:

- The former **Google** (KML) output option has been been moved to the dedicated Export Google Earth File command.

Prompts

Polyline file format [<Carlson>/DTM/Idan/MicroStation/MOSS/Peabody/Topcon/12D]? *Specify the desired output option by specifying the CAPITALIZED option or press Enter for the <default> option.*

Polyline File to Write dialog: Create a new file or Append to Existing. If the **Carlson** option was selected, the following dialog then appears:



Use Polyline File for Grid File Utilities macro: When enabled, the option will write a polyline file that can be used with Grid File Utilities for inclusion/exclusion perimeters.

Specify Exclusion/Warning Polylines: When enabled, this option applies to machine control for warning areas.

Specify WorkZone Polylines: When enabled, this option applies to machine control for working areas.

Reduce Polyline Vertices: When enabled, this option applies the Reduce Polyline Vertices to the polyline vertices before writing the file..

Offset Cutoff: Indicate the allowable offset distance (essentially the *middle ordinate distance* of a 3-point arc) that would allow the middle vertex between two other vertex locations to be removed.

Include Z coordinate in polyline file: When enabled, this option controls whether the elevation(s) (or "Z" value) of the selected polyline vertices are written to the polyline file.

Decimals: Indicate the desired amount of precision for the coordinate values that should be written to the file.

Select polylines, lines and arcs to write.
FILter/<Select entities>: *Pick the entities to process press Enter when complete.*

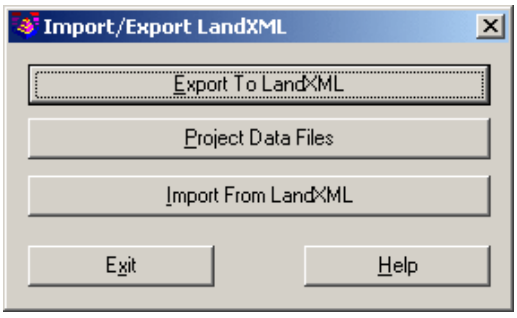
Sample Polyline File:

```
POLYLINE,51,0,0.0,CONT|V-STRM-PIPE
5375168.9320,3932304.7050,0.0000
5375193.3310,3932211.6150,0.0000
POLYLINE,150,0,0.0,CONT|V-BRKL
5375026.8800,3932090.0480,962.8334
5375062.3960,3932105.7540,961.5399
5375075.5640,3932115.7940,961.1595
5375079.0150,3932128.0920,961.1532
5375081.6860,3932159.7840,961.6147
5375086.6920,3932195.6480,962.6206
etc.
```

Pulldown Menu Location: File > Export
Keyboard Command: polywrite
Prerequisite: Polylines in the drawing

Export LandXML File

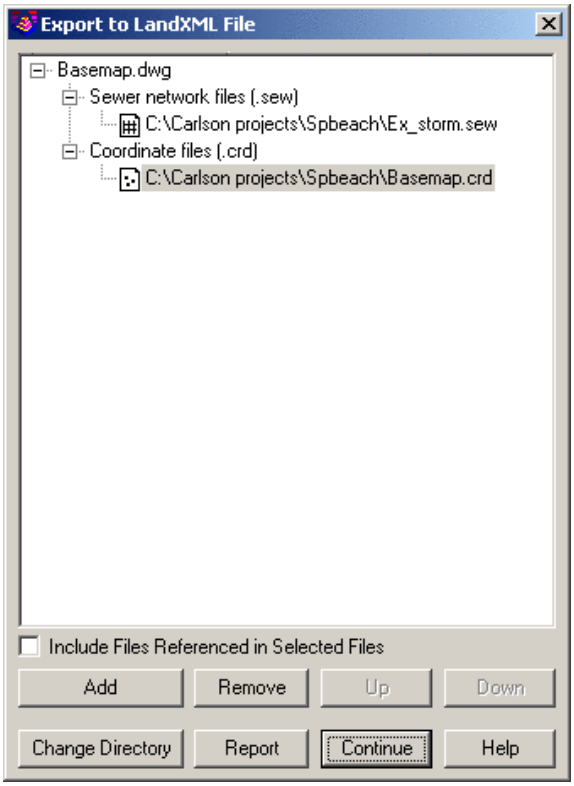
The Export LandXML File routine provides a mechanism where data can be sent from Carlson Software into a LandXML file for use in other applications that support the LandXML data specification. To generate a LandXML file, a series of dialog boxes are presented:



Export to LandXML: This option allows you to individually select the desired Carlson Software data file(s) that should be included in the LandXML file.

Project Data Files: This option allows you to quickly select the various data files associated with, and defined by a Carlson Project (∗.prj) file.

Select LandXML File: Specify the name of a LandXML file you wish to create.

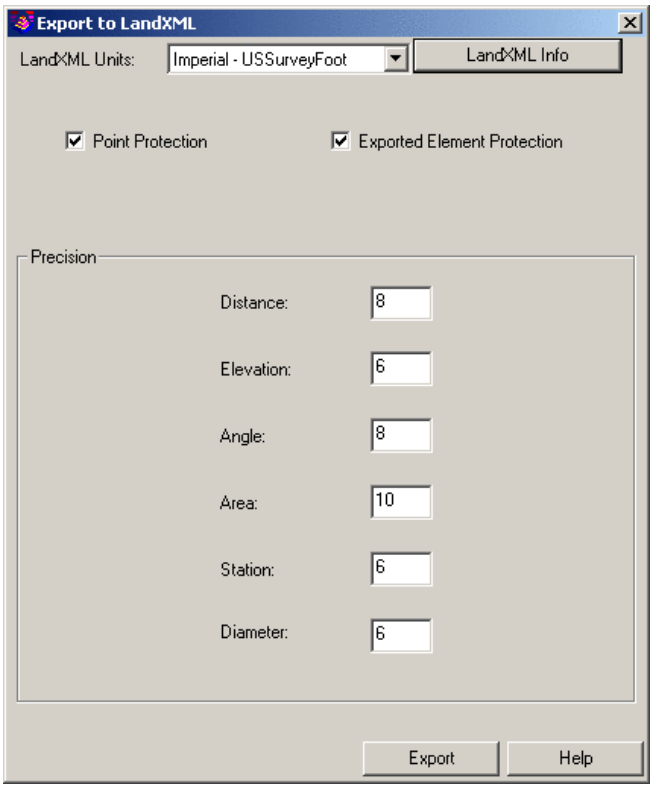


Include Files Referenced in Select Files: When enabled, this option will automatically add other files that are referenced by the selected file. As an example, the file produced by the Carlson Road Network command references TINs, Centerlines, Profiles, *etc*, and adding the single Road Network file will also add the referenced file(s) into the Export to LandXML File dialog box.

Export to LandXML File: Add, remove (using standard Windows **click**, **shift+click** and/or **ctrl+click** functionality) or otherwise organize the data file(s) that is to be incorporated into the LandXML file.

Change Directory: This option allows you to adjust the folder location from where selected data files should be referenced (often used for project revision purposes).

Report: Create a report (suitable for file transmission or archival purposes) of the file(s) selected to be incorporated into the LandXML file.



LandXML Units: Specify the desired Units of Measure that reflect the outgoing data.

Point Protection: When enabled, you are prompted for a course of action if an existing LandXML file you've selected contains COGO points that have the same number(s) as those being selected for the LandXML file. When disabled, point data you've selected for the LandXML file are automatically written to (or updated into) the existing LandXML file.

Exported Element Protection: When enabled, you are prompted if existing data (such as a centerline) in a LandXML file should be updated with data of the same name that you have selected for the LandXML file.

Precision: Set the desired level of precision for each of the various measurement categories.

Profiles:

There are two major different types of profiles in LandXML: ProfSurf and ProfAlign. ProfSurf is typically an existing surface that is usually created using existing surface data. The data for this type of profile it is stored in a series of station-elevation values as a representation of a PntList2D list. ProfAlign is for a design profile. The data for this type of profile is stored in LandXML elements starting from the simplest one: PVI element, CircCurve element, ParaCurve element, etc.

Carlson differentiates the two types mentioned above by using the profile type in the Carlson .pro file: Generic = ProSurf, Road = ProAlign.

Note:

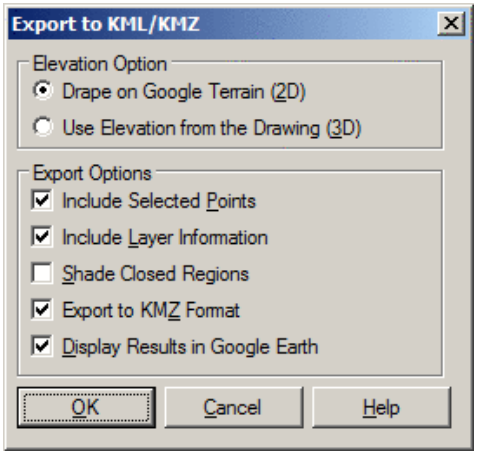
- The LandXML initiative is being driven by the land development industry as an acceptable means to share and transfer land data rather than the traditional graphical representation of that data. It also provides an effective means for transferring a variety data (points, centerlines, profiles, surface models, sewer data, *etc*). Another

advantage of LandXML is that the LandXML data structure is CAD and software vendor neutral (meaning you don't have to own or use the CAD or software product used by your data provider).
- Visit http://www.landxml.org for additional information on the uses and acceptance of the LandXML initiative.

Pulldown Menu Location(s): File > Export
Keyboard Command: landxml_export
Prerequisite: Carlson project data files to convert

Export Google Earth File

The Export Google Earth File allows you to produce a KML (Keyhole Markup Language or alternatively a KMZ) file of Carlson points, lines, arcs and polylines for rendering in other mapping and GIS applications such as Google Earth. Throughout this discussion, *KML* will be used to also describe KMZ files unless explicitly noted.



Drape on Google Terrain (2D): When this option is selected, entities written to the KML file will have an Altitude setting of "Clamped to ground."

Use Elevations from the Drawing (3D): When this option is selected, entities written to the KML file will have an Altitude setting of "Absolute."

Include Selected Points: When enabled, this option exports selected Carlson point information to the KML <Placemark><Point>...</Point></Placemark> tag structure.

Include Layer Information: When enabled, this option organizes exported information based on the layer of each entity, with each CAD layer becoming a KML <Folder>...</Folder> entry with the color of the group taking the general color of the CAD layer.

Shade Closed Regions: When enabled, all closed polyline regions (*e.g.* building pads, ponds, *etc*) will be fill-shaded.

Export to KMZ Format: When enabled, the KML file is written to the more compact (zipped) KMZ version of the standard KML file format.

Display Results in Google Earth: When enabled, the results of the KML are passed to and automatically opened with Google Earth.

Note:

- When the *Use Elevations from the Drawing (3D)* option is selected, be aware that elevation values lower than the Google Earth terrain may be obstructed in the Google Earth display.
- Attribute information (*e.g.* Number, Elevation, Description) of selected Carlson points are also written to the KML and will display in the "balloon" when a point is picked in the Google Earth display or data hierarchy.

- When the *Shade Closed Regions* toggle is enabled, note that **all** closed polyline regions will become fill shaded and may lead to undesired results for items such as closed contours.
- When prompted for the name of the KML/KMZ file to write, the appropriate KML or KMZ file extension based on the *Export to KMZ Format* toggle will be added to the file if the file extension is not specified.
- Arcs and polylines with arcs are converted into chord segments that closely approximate the arc(s).
- Other entities not supported for direct export to a KML file (*e.g.* circles, 3DFaces, ellipses, splines, multilines, regions and solids), can be first turned into polylines with the Entities to Polylines command. Text entities can be converted to polylines through the use of the Text Explode To Polylines command.
- The graphical symbology of any/all items sent to the KML file can be manually modified via the Google Earth interface.

Prompts

Select points, polylines, lines and arcs to write.
FILter/<Select entities>: *Select the desired entities and press Enter when complete.*

- To import a Google Earth image into your drawing, use the Place Google Earth Image command.
- To import a Google Earth terrain data into a Carlson TIN (surface model), use the Place Google Earth Image command.
- To import KML content into your drawing, use the Import Google Earth File command.

Pulldown Menu Location: File > Export
Keyboard Command: kmlwrite
Prerequisite: Points, lines or polylines in the drawing with an established projection zone through Drawing Setup.

Exit

This command allows you to exit AgStar.

This option allows you to quit the program if there have been no changes since the drawing was last saved. If the drawing has been modified, the program displays the Drawing Modification dialog box to prompt you to save or discard the changes before quitting.

You can close a file that has been opened in read-only mode if you have made no modifications or if you are willing to discard them. To save modifications to a read-only drawing, use the SAVEAS command under the File menu.

Menu Location: File
Prerequisite: None
Keyboard Command: QUIT

Edit Commands 3

Undo

This command allows you to reverse the effect of previously issued commands.

Menu Location: Edit
Prerequisite: None
Keyboard Command: U

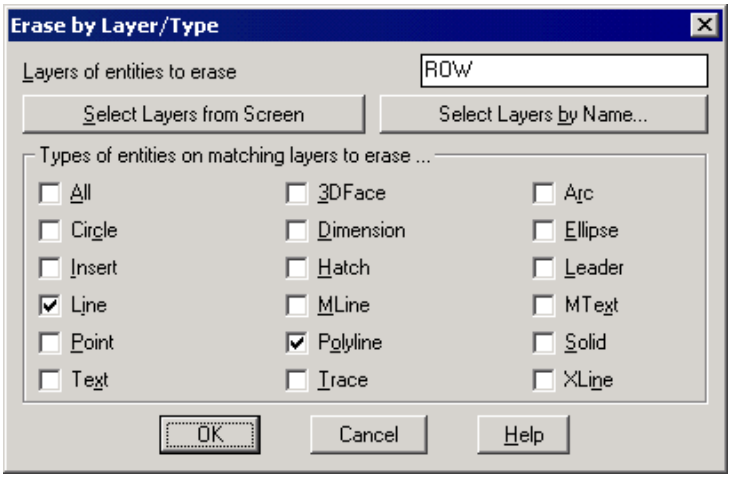
Erase

This command allows you to remove objects from a drawing.

Menu Location: Edit
Prerequisite: None
Keyboard Command: ERASE, E

Delete Layer

This command erases all the entities on the specified layers and deletes the layers from the drawing.



1 In the Erase by Layer/Type dialog box, you can enter the name of the layer to be deleted, or you can specify a layer to delete by picking an entity on that layer.

- **Layers of entities to erase**: This field displays the layer names to be erased. You also have the option of typing layer names in this field.
- **Select Layers from Screen**: This option allows you to select entities on the layers you want deleted.
- **Select Layers by Name**: This option allows you to choose a layer name from the list of layers in the drawing.

2 Under Types of entities on matching layers to erase, you specify which types of entities to erase.

You can clear All and specify which entity types to include in the selection set. For instance, if you have both linework and points on the same layer, and you want to erase only the linework, you can clear All and select Line and Polyline.

Menu Location: Edit
Prerequisite: None
Keyboard Command: LDEL

Move

This command allows you to displace objects a specified distance in a specified direction.

Menu Location: Edit
Prerequisite: None
Keyboard Command: MOVE, M

Copy

This command copies all objects you select to the Clipboard. You can paste the contents of the Clipboard into a document or drawing as an OLE object.

You can also use CTRL+C to run this command. If the cursor is in the drawing area, Carlson Survey copies the selected objects to the Clipboard. If the cursor is on the command line or in the text window, the program copies the selected text to the Clipboard.

Menu Location: Edit, Clipboard >
Prerequisite: None
Keyboard Command: COPYCLIP

Explode

This command allows you to break a compound object into its component objects.

Results differ depending on the type of compound object you're exploding. The following is a list of objects that can be exploded and the results for each.

- **All Explodable Objects**: Produces object geometry that may look the same, but the color, linetype, and lineweight of the object may change.
- **Block**: Removes one grouping level at a time. If a block contains a polyline or a nested block, exploding the block exposes the polyline or nested block object, which must then be exploded to expose its individual objects.

Blocks with equal X, Y, and Z scales explode into their component objects. Blocks with unequal X, Y, and Z scales (nonuniformly scaled blocks) might explode into unexpected objects.

When nonuniformly scaled blocks contain objects that cannot be exploded, they are collected into an anonymous block (named with a "∗E" prefix) and referenced with the nonuniform scaling. If all the objects in such a block cannot be exploded, the selected block reference will not be exploded. Body, 3D Solid, and Region entities in a nonuniformly scaled block cannot be exploded.

Exploding a block that contains attributes deletes the attribute values and redisplays the attribute definitions.

- **2D and Lightweight Polyline**: Discards any associated width or tangent information.
- **Wide Polyline**: Places the resulting lines and arcs along the center of the polyline. Carlson Survey discards any associated width or tangent information.
- **3D Polyline**: Explodes into line segments. Any linetype assigned to the 3D polyline is applied to each resulting line segment.
- **Leaders**: Explodes into lines, splines, solids (arrow heads), block inserts (arrow heads, annotation blocks), Mtext, or tolerance objects, depending on the leader.
- **Mtext**: Explodes into text entities
- **Multiline**: Explodes into lines and arcs.
- **3D Solid**: Explodes planar surfaces into regions. Nonplanar surfaces explode into bodies.
- **Region**: Explodes into lines, arcs, or splines.
- **Body**: Explodes into a single-surface body (nonplanar surfaces), regions, or curves.
- **Polyface Mesh**: Explodes one-vertex meshes into a point object. Two-vertex meshes explode into a line. Three-vertex meshes explode into 3D faces.

- **Circle Within a Nonuniformly Scaled Block**: Explodes a circle within a nonuniformly scaled block into ellipses.
- **Arc Within a Nonuniformly Scaled Block**: Explodes an arc within a nonuniformly scaled block into elliptical arcs.

Menu Location: Edit
Prerequisite: None
Keyboard Command: EXPLODE, X

Offset

This command creates a new object at a specified distance from an existing object or through a specified point. Offset does not support 3D polylines. Use Offset 3D Polyline to offset these entities.

Prompts

1 Specify offset distance or [Through] <Through>: **Press Enter**

The Through option allows you to screen pick the location of the offset. You can also enter a value for the interval of the offset.

2 Select object to offset or <exit>: **select entity**

3 Specify through point: **pick point**

Menu Location: Edit
Prerequisite: None
Keyboard Command: OFFSET

Trim

This command allows you to trim objects at a cutting edge defined by other objects.

Prompts

1 Select cutting edges ...

Select objects: **pick entity**

2 Select object to trim or shift-select to extend or [Project/Edge/Undo]: **select entity to be trimmed**

- **Project**: You can project the object to be trimmed in order to trim objects that do not intersect.
- **Edge**: You can project the trimming edge in order to trim objects that do not intersect.
- **Undo**: This option allows you to undo the above projections.

Menu Location: Edit
Prerequisite: None
Keyboard Command: TRIM, TR

Scale

This command allows you to enlarge or reduce selected objects equally in the X, Y, and Z directions.

Prompts

1 Select objects: **pick entities**

2 Specify base point: pick point on screen as reference

3 Specify scale factor or [Reference]: **scale to desired size**

Menu Location: Edit
Prerequisite: None
Keyboard Command: SC

Extend To Edge

This command allows you to extend an object to meet another object.

Prompts

1 Select boundary edges ...

Select objects: **pick entity**

2 Select object to extend or shift-select to trim or [Project/Edge/Undo]: **pick entity**

You have the option of trimming or projecting objects and edges.

Menu Location: Edit, Extend >
Prerequisite: None
Keyboard Command: EXTEND

Extend by Distance

This command extends a line or polyline, or creates a new line or polyline from an existing one, using a specified distance. The new segment of the line or polyline is drawn from the current position in the direction indicated by the current position arrowhead.

You start by selecting an existing line or polyline. Initially, the current position is the closest vertex to the location where the line or polyline was selected. Extending from the endpoint of a polyline will add a new point to that polyline, while extending from any other point will create a new polyline.

There are two modes of operation: draw mode (D) and move mode (M). When you are in draw mode, extending will draw line or polyline segments. When you are in move mode, the current position arrowhead can be moved without drawing segments. The orientation of the current position arrowhead can be changed with the Right, Left, and Angle commands.

Here is a list of the Extend by Distance options:

Key Name Action

D	Draw mode	Actions draw or extend the line or polyline
M	Move mode	Actions only move the pointer
#	Number	Distance to draw or extend
R	Right rotate	Rotates clockwise 90 degrees
L	Left rotate	Rotates counterclockwise 90 degrees
E	Extend to edge	Extends to intersection with a selected line or polyline
T#	Total distance	Sets current segment to specified distance
A#	Angle change	Rotates pointer by specified number of degrees
A	Align	Rotates pointer to align with segment

B	Bearing	Sets pointer direction by bearing in format: Qdd.mmss with Q-quadrant, d-degrees, m-minutes, s-seconds (e.g. 130.1005 is NE 30 degrees, 10 minutes, and 5 seconds)
S	Switch	Reverses pointer direction
N	Next	Moves pointer to next point
P	Previous	Moves pointer to previous point
U	Undo	Undo the last Extend by Distance command
C	Close	Closes the polyline
O	Open	Opens the polyline
Help	Help	Displays this list

Prompts

1 Select line or pline to extend: **select line or polyline**

Pick the polyline near the place to extend

2 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **T50**

The line is extended to a total length of 50 units.

3 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **R**

The pointer is turned to the right.

4 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **75**

A polyline is drawn to 75 units.

5 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **B145.0000**

The pointer is turned toward NE at a bearing of 45 degrees.

6 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **50**

A polyline is drawn to 50 units.

7 Enter or pick distance to draw (A,B,C,E,L,M,N,O,P,R,S,T,U,Help): **Press Enter**

Press Enter to end the command.

Menu Location: Edit, Extend >
Prerequisite: An existing line or polyline with at least one segment from which to start.
Keyboard Command: EXTENDER

Break by Closed Polyline

This command will force a break where all lines and polylines cross the "break" polyline. In addition to breaking lines and polylines, you can also put the segments to new layers based on whether the segments are inside or outside the break polyline. Polylines with arc segments are not handled. To reconnect the broken lines and polylines, use the Join Nearest command.

Prompts

1 Select the clip edge polyline: **pick a closed polyline**

2 Select the polylines and lines to be clipped.

Select Objects: **pick the entities to break**

3 Specify layer names for Inside segments (Yes/<No>)? **Yes**

4 Enter a layer name for the Inside segments <0>: **Press Enter**

5 Specify layer names for Outside segments (Yes/<No>)? **Yes**

6 Enter a layer name for the Outside segments <0>: **Final**

Menu Location: Edit, Break >
Prerequisite: A closed polyline
Keyboard Command: CLIPLINE

Break at Intersection

This command allows you to break a line, arc, or polyline at the intersection of another line, arc, or polyline. This command is often used in conjunction with the Area by Lines & Arcs command because many times, to get the correct area of a figure, you must break it from adjoining lines.

Prompts

1 Select Line, Arc, or Polyline to Break

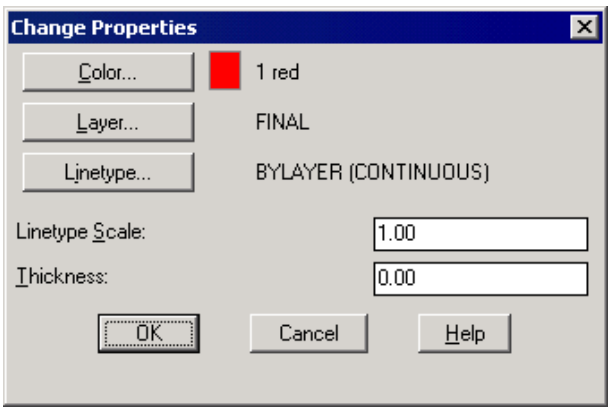
Select object: **select entity to break**

2 [int on] Pick Intersection to break at: **pick intersection point**

Menu Location: Edit, Break >
Prerequisite: None
Keyboard Command: BREAKAT

Change Properties

This command allows you to change certain properties of existing objects.



1 In the Change Properties dialog box, you must choose the properties to modify.

- **Color**: This option allows you to change the color of the object.
- **Layer**: This option allows you to change the layer of the object.
- **Linetype**: This option allows you to change the linetype of the object.
- **Linetype Scale**: This option specifies the linetype scale factor for the new linetype.
- **Thickness**: This option specifies the distance to extrude the object above or below its elevation.

Note: The Properties command allows you to modify entity specific properties such as the radius of a circle or the height of a text entity.

Menu Location: Edit, Change >
Prerequisite: None
Keyboard Command: DDCHPROP

Change Elevations

This command allows you to change the elevation of selected entities. You can move the entity to a specified elevation from its current elevation (absolute) or you can specify a differential change to be added to or subtracted from an entity's current elevation. If AgStar points are selected, their attribute text and Z axis (Elevation) coordinate are changed. Elevation values are not changed in the current coordinate file. You must choose the command Update CRD File from Drawing under Coordinate File Utilities on the Points menu to update the elevation values in the current coordinate file.

Prompts

1 Ignore zero elevations (<Yes>/No)? **Press Enter**

If you answer No, then entities with elevation 0 will be changed.

2 [A]bsolute or [D]ifferential Change <A>: **A**

3 Elevation to change to <0.0>: **125**

4 Select Entities for elevation change.

Select objects: **pick entities from screen**

5 Select objects: **Press Enter**

Menu Location: Edit, Change >
Prerequisite: an entity with elevation
Keyboard Command: CHGELEV

Rotate

This command changes the display angle of objects by rotating them. This command has a sub-menu consisting of two items, as described below :

By Bearing

Function

This command allows the user to rotate the coordinate database of all or part of the drawing. To translate the coordinate database, use the *MOVE* command.

Prompts

Select entities to rotate. Select objects: *W*

First corner: *(pick point)*

Other corner: *(pick point)*

Select objects: *[Enter]*

Base pivot point ?

Number/<Pick point>: *N*

By responding with N the program switches to point number input.

Pick point/<point number>: *2*

The program then reads the coordinate value from the current CooRDinate file.

Reference Bearing point ?

Pick point/<point number>: *P*

By responding with P the program then switches back to screen picking mode.

Number/<Pick point:>: *(pick point)*

Reference Bearing N 44d31'1" E

The program then displays the reference bearing defined by the two points selected.

Azimuth/<Bearing (Qdd.mmss)>: *245.3030*

Enter an A to input an Azimuth or enter the bearing. The above response defines a bearing of South 45 degrees, 30 minutes, and 30 seconds East.

The program then changes the database to the new bearing.

If point objects are selected the program warns:

This command DOES NOT change the coordinates in the CooRDinate file!

Use CooRDinate File Utilities menu option F to update the file.

Standard Rotate

Function

This command allows the user to choose objects and then prompts for a base point. It then prompts the user to choose whether to rotate by rotation angle (default) or by reference angle. As the mouse is moved around, the rotated position of the object keeps changing, giving an idea

of its final position. The user can then click on a point to select the angle of rotation and thereby, the final position of the object. The angle of rotation can be also be entered from the keyboard. The object is rotated through that angle, relative to its current inclination to the coordinate axes. Whether the rotation is clockwise or counter-clockwise can be set using the "Units Control" command under the "Inq-Set" pulldown menu. Alternatively, the rotation can be achieved by reference angle. This is done by first choosing the "Reference" option after entering the base point. The user then has to select the reference angle, whose orientation needs to be changed. The user can either enter this at the keyboard or choose a line on the object by selecting two points on it. The user is then prompted to choose the new angle, which can again be entered at the keyboard or selected by choosing two points on an exisiting object. This option of rotating by reference is particularly useful when objects need to be aligned by rotation.

Prompts

Select objects: *pick an objects to be rotated (press <ENTER> after selecting required objects, to end selection)*

Base point : *pick base point*

<Rotation Angle>/Reference : *40 (objects will be rotated by 40 units about the base point, the units and direction depending on the settings in the Units Control dialog box.)*

<Rotation Angle>/Reference : *type 'ref' to choose rotation by reference angle*

Reference angle <E> : *Choose the direction which is meant to be the reference angle. For example, if a line at 45^o to the horizontal needs to be at 120^o in the resultant figure, then, the reference angle needs to be chosen as 45^o . This can be done in two ways : typing 45^o at the keyboard or choosing two points on the reference line which is oriented in the required angle (45^o in this case.)*

New angle : *Choose the direction that the reference angle previously chosen needs to be oriented. In the example under consideration, the new angle will be 120^o . Once again, it can be specified in two ways : from the keyboard or by specifying two points on the drawing.*

Edit Text

This command allows you to edit text and attribute labels.

1 Select Text to Edit: **select the text**

You can modify text in provided text field.

Menu Location: Edit, Text>
Prerequisite: Text
Keyboard Command: EDITXT

Text Style

This command allows the user to change the current text style to one of the available text styles. The font style can be given any name by the user, following which the user is prompted to select the specific font file to be used and the text orientation, inclination, height, width etc. If the "?" option is entered, the current text style and its details are listed.

Pull-Down Menu Location: Edit
Prerequisite: None.

Text EnlargeReduce

This command allows you to scale text entities up or down in size. The command prompts for a scale multiplier and a selection set of text objects. If you want to enlarge the text, enter a value greater than one. If you want to reduce the text, enter a decimal fraction such as .5, which would reduce the text size by 50%. Use this command if you set up your drawing for one plotting scale and then decide to change to a new plotting scale. The Change Text Size command can alternatively be used to set the text size to a specific value.

Prompts

1 Scaling Multiplier: **enter value**

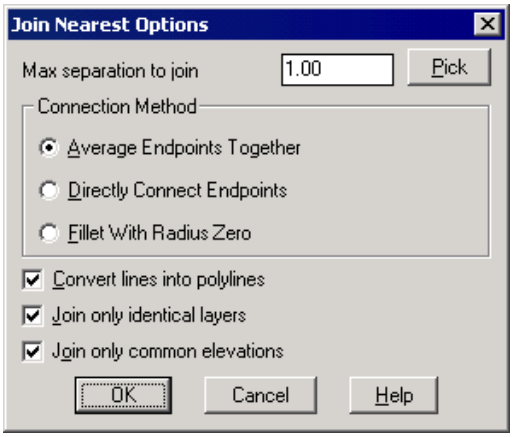
Select Text for Scaling.

2 Select objects: **select the text**

Menu Location: Edit, Text >
Prerequisite: Text
Keyboard Command: TXTENL

Join Nearest

This command joins lines or polylines together, and allows you to join lines that do not exactly meet. You specify the maximum distance to join, along with other options, in the dialog box shown below. You can join many entities at once.



1 You must specify the maximum separation distance parameter. Entities beyond this distance will not join.

2 Under Connection Method you must determine how entities are connected.

- **Average Endpoints Together**: This option averages together the endpoints of the two entities when joined.
- **Directly Connect Endpoints**: This option directly connects the endpoints of the two entities with a polyline.
- **Fillet With Radius Zero**: This option will perform a tight fillet on the two entities.

3 In the Join Nearest Options dialog box you can choose to join only lines with common elevations or layers.

- **Convert lines into polylines**: The option will automatically convert any lines in the selection set into polylines.
- **Join only identical layers**: This option will join entities on the same layer.
- **Join only common elevations**: This option will join entities with identical elevations.

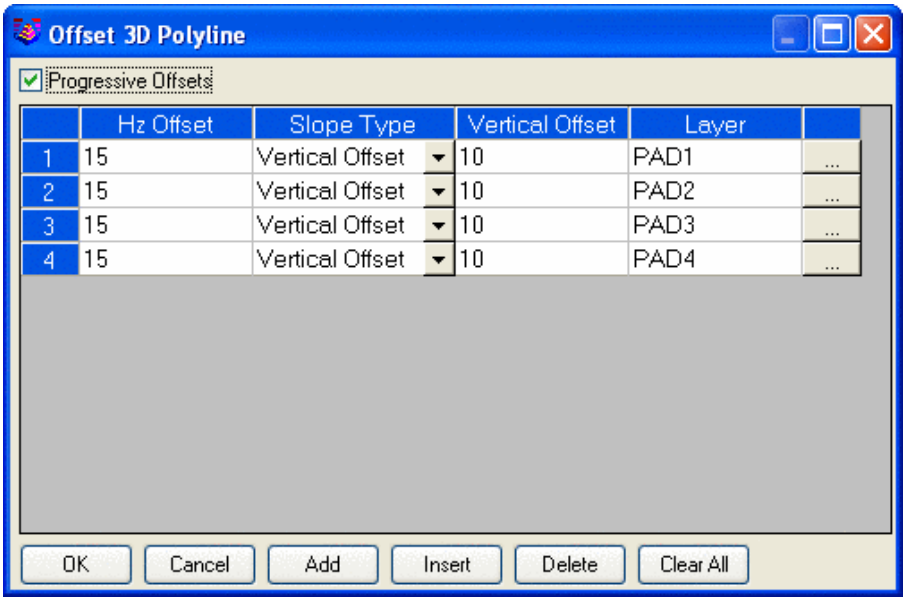
Menu Location: Edit
Prerequisite: Lines or Polylines to be joined.
Keyboard Command: NEARJOIN

Offset 3D Polyline

This command allows you to offset a 3D polyline entity in both the horizontal and vertical directions. There are five offset methods. The Interval method applies one horizontal and one vertical offset to all the vertices of the polyline. The Constant method has a horizontal offset and sets the elevation of the polyline to one constant elevation. The Variable method allows you to specify each horizontal and vertical offset individually either by polyline segment or for each point. The vertical offset can be specified by actual vertical distance, percent slope or slope ratio.

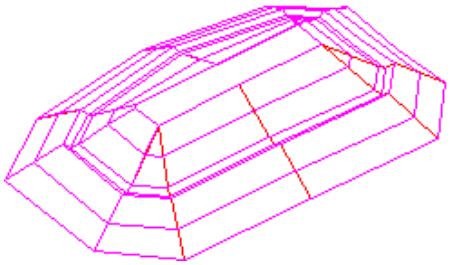
The surface method allows to offset/project a 3D polyline entity on to a surface (tin;flt;grd) based on cut and fill outslope ratio.

The multiple method allows multiple offsets of a 3D polyline with separate layers. User can add, insert and delete offsets rows and set individual layers. The option Progressive Offsets draws offsets progressively, i.e. successive offsets uses last drawn offset as base.



Prompts

Enter the offset method [<Interval>/Constant/Variable/Surface/Multiple]: *press Enter*
Vertical/<Horizontal offset amount>: *15*
Percent/Ratio/Vertical offset amount <0>: *10*
Select a polyline to offset (Enter for none): *select a 3D poly*
Select side to offset: *pick a point*
Select a point on the graphics screen that is in the direction of the side of line to offset.
Select a polyline to offset (Enter for none): *press Enter*



Pulldown Menu Location: Edit > Polyline Utilities
Keyboard Command: offset3d
Prerequisite: polyline

Entities to Polylines

This command converts selected lines, arcs, circles, 3Dfaces, and solids into individual polylines. You may use Join Nearest to convert adjoining lines and arcs into continuous polylines.

Prompts

1 Select lines, arcs, circles, 3Dfaces and solids to convert.

Select objects: **pick entities**

Menu Location: Edit, Polyline Utilities>
Prerequisite: Lines, arcs or other entities to convert.

Keyboard Command: TOPLINE

Reverse Polyline

This command reverses the order of the line and/or arc segments of a polyline. This can be used in conjunction with the commands such as Station Polyline/Centerline or Profile from Surface Entities, since the polyline must be plotted in the direction of increasing stations. If it is more convenient to draft a polyline in one direction you may do so and then use the Reverse Polyline command to change its order. This command can also be used to reverse a 3D Polyline Breakline or a 3D Pad Template. Temporary arrows are drawn along the polyline to graphically show the new polyline direction.

Prompts

1 Select the Polyline to Reverse: **pick point on polyline**

Menu Location: Edit, Polyline Utilities>
Prerequisite: A polyline
Keyboard Command: REVPLINE

Reduce Polyline Vertices

This command removes points from a polyline without significantly changing the polyline. The offset cutoff is the maximum distance that the polyline can move when you remove a point. For example, in a polyline with three points in a straight line, the middle point can be removed without changing the polyline.

Prompts

1 Enter the offset cutoff <0.1>: **.5**

2 Select polylines to reduce.

Select objects: **pick polylines**

Menu Location: Edit, Polyline Utilities>
Prerequisite: A polyline
Keyboard Command: REDUCE

Smooth Polyline

This command allows you to smooth selected polylines using a modified Bezier method. The resulting polyline passes through all the original points, and only the segments between the original points are smoothed. The looping factor controls the smoothing amount. A higher factor gives more looping.

Prompts

1 Enter the looping factor (1-10) <5>: **7**

2 Enter the offset cutoff <0.05>: **Press Enter**

3 Select polylines to smooth.

Select objects: **pick polylines**

Menu Location: Edit, Polyline Utilities>
Prerequisite: A polyline
Keyboard Command: SMOOTHPL

Add Polyline Vertex

This command allows you to add points to a polyline. Select the polyline to modify, then pick or enter the coordinates for the new point. The new point is inserted into the polyline at the nearest polyline segment. On a 3D polyline, the elevation of the new vertex is calculated for you.

Prompts

1 Select polyline to add to: **pick a polyline**

2 Pick or enter point to add: **pick a point**

Menu Location: Edit, Polyline Utilities>
Prerequisite: A polyline
Keyboard Command: ADDPL

Close Polylines

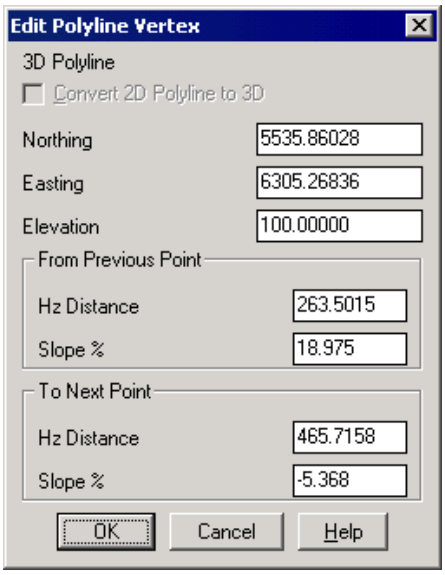
This command allows you to close a selection set of open polylines.

Menu Location: Edit, Polyline Utilities>
Prerequisite: Open polyline(s).
Keyboard Command: CLOSEPL

Edit Polyline Vertex

This tool allows you to make changes in the coordinates of vertices on all polyline types. Upon execution you will be asked to select a polyline to edit. Upon selection a temporary marker will be placed at all of the vertices of the polyline, making them easy to distinguish. Then pick near the vertex you wish to edit, and the following dialog appears.

At the top of the dialog it identifies the type of polyline, being 2D or 3D. In the case of 2D polylines it allows you convert the polyine. You have the ability to type in new northing, easting or elevation values. You can also determine the 3D coordinate position by using distances and slope to/from adjacent points. As you change the values in the dialog, new values for derivatives are being calculated. For example if you change the horizontal distances, the coordinates will change.



Prompts

1 Select polyline to edit: pick a 2D or 3D polyline

After you pick a polyline, the vertices are marked with a temporary X symbol

2 Pick point on polyline to edit: **pick the vertex to edit**

3 Dialog box is shown

4 Select polyline vertex to edit: press enter to quit or select another polyline

Menu Location: Edit, Polyline Utilities>
Prerequisite: A polyline
Keyboard Command: EDITPL

Open Polylines

This command allows you to open a selection set of closed polylines.

Menu Location: Edit, Polyline Utilities>
Prerequisite: Closed polyline(s).
Keyboard Command: OPENPL

Remove Polyline Arcs

This command allows you to replace arc segments in polylines with chords. Removing arcs is a prerequisite to some commands that don't handle arcs, such as Break by Closed Polyline and Make 3D Grid file. This command can add many vertices to the polyline.

Prompts

1 Select polylines to remove arcs from.

Select objects: **pick polylines**

2 Offset cutoff <0.5>: **Press Enter**

This specifies the maximum distance that any point on the arc will be allowed to shift.

Menu Location: Edit, Polyline Utilities>
Prerequisite: A polyline
Keyboard Command: RMARC

Remove Polyline Segment

This command allows you to remove a specified segment from a polyline. (A polyline segment is the section between two vertices of the polyline.) There are two options for removing the segment. When you specify the Continuous option, the two vertices of the removed segments are averaged together to keep the polyline continuous. When you specify the Break option, the segment is left missing in the polyline, resulting in two separate polylines.

Prompts

1 Break polyline at removal or keep continuous [<Break>/Continuous]? **C**

2 Select polyline segment to remove: **pick point on polyline segment**

Menu Location: Edit, Polyline Utilities>
Prerequisite: A polyline
Keyboard Command: REMOVEPL

Remove Polyline Vertex

This command allows you to remove the selected vertex from a polyline.

Prompts

1 Select polyline vertex to remove: **pick point on polyline**

Select the vertex to remove

Menu Location: Edit, Polyline Utilities>
Prerequisite: A polyline
Keyboard Command: RMVERTEX

Image Clip

This command allows you to create new clipping boundaries for an image object.

Prompts

1 Select image to clip: select the edge of an image

2 Enter image clipping option [ON/OFF/Delete/New boundary] <New>: **enter an option or Press Enter**

The boundary you specify must be in a plane parallel to the image object.

- **On**: Turns on clipping and displays the image clipped to the previously defined boundary.
- **Off**: Turns off clipping and displays the entire image and frame. If you reclip the image while clipping is turned off, the program automatically turns clipping back on. The program prompts you to delete the old boundary even when clipping is turned off and the clipping boundary is not visible.
- **Delete**: Removes a predefined clipping boundary and redisplays the full original image.
- **New Boundary**: Specifies a new clipping boundary. The boundary can be rectangular or polygonal, and consists only of straight line segments. When defining a clipping boundary, specify vertices within the image boundary. Self-intersecting vertices are valid. Rectangular is the default option. If you use the pointing device to specify a point at the Enter Clipping Type prompt, the program interprets the point as the first corner of a rectangle.

3 Enter clipping type [Polygonal/Rectangular] <Rectangular>: **enter P or Press Enter**

- **Polygonal**: Uses specified points to define a polygonal boundary.

Specify first point: Specify a point

Specify next point or [Undo]: **specify a point or enter u**

Specify next point or [Undo]: **specify a point or enter u**

Specify next point or [Close/Undo]: **specify a point, or enter c or u**

You must specify at least three points to define a polygon.

If the image already has a clipping boundary defined, AgStar displays the following prompt:

Delete old boundary? [No/Yes] <Yes>: **enter N or Press Enter**

If you choose Yes, the program redraws the entire image and the command continues; if you choose No, the command ends.

- **Rectangular**: Specifies a rectangular boundary by its opposite corners. AgStar always draws the rectangle parallel to the edges of the image.

Specify first corner point: **specify a point**

Specify opposite corner point: **specify a point**

Menu Location: Edit, Image>
Prerequisite: None
Keyboard Command: IMAGECLIP

Image Frame

This command controls whether Carlson Survey displays the image frame or hides it from view.

Because you select an image by clicking its frame, setting the image frame to off prevents you from selecting an image.

Prompts

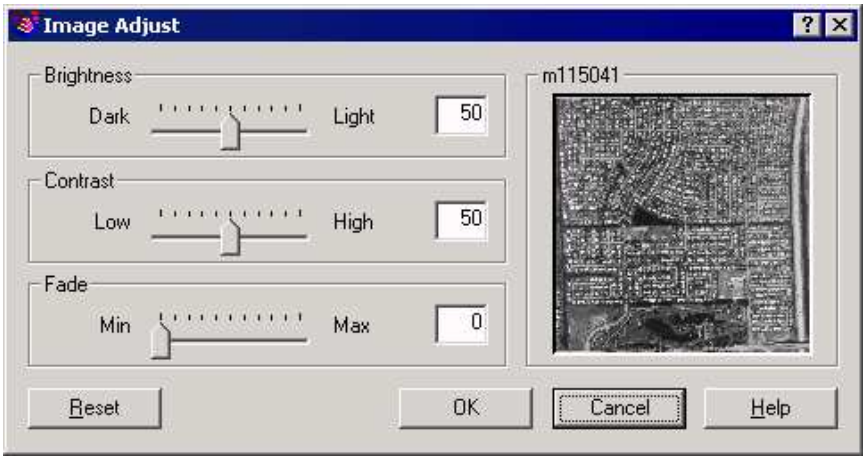
1 Enter image frame setting [ON/OFF] <current>: **enter an option or Press Enter**

- **On**: Displays image frames so you can select images.
- **Off**: Hides image frames so you cannot select images.

Menu Location: Edit, Image>
Prerequisite: None
Keyboard Command: IMAGEFRAME

Image Adjust

This command controls the display of the brightness, contrast, and fade values of images.



The Image Adjust dialog box controls how the image is displayed by adjusting the brightness, contrast, and fade settings of the selected image. Adjusting these values changes the display of the image but does not change the image file itself.

- **Brightness**: Controls the brightness, and indirectly the contrast, of the image. Values range from 0 through 100. The greater the value, the brighter the image and the more pixels that become white when you increase contrast. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

- **Contrast**: Controls the contrast, and indirectly the fading effect, of the image. Values range from 0 through 100. The greater the value, the more each pixel is forced to its primary or secondary color. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

- **Fade**: Controls the fading effect of the image. Values range from 0 through 100. The greater the value, the more the image blends with the current background color. A value of 100 blends the image completely into the background. Changing the screen background color causes the image to fade to the new color. In plotting, the background color for fade is white. Moving the slider to the left decreases the value; moving the slider to the right increases the value.

- **Image Preview**: Displays a preview of the selected image. The preview image updates dynamically to reflect changes to the brightness, contrast, and fade settings.
- **Reset**: Resets values for brightness, contrast, and fade to default settings (50, 50, and 0, respectively).

Menu Location: Edit, Image>
Prerequisite: None
Keyboard Command: IMAGEADJUST

View Commands 4

Zoom - Window

This commands zooms to display an area you specify by two opposite corners of a rectangular window.

Menu Location: View, Zoom>
Prerequisite: None
Keyboard Command: ZOOM, W

Zoom Previous

This command zooms to display a previous view. You can restore up to 10 previous views.

Menu Location: View, Zoom>
Prerequisite: None
Keyboard Command: ZOOM, P

Zoom Center

This command zooms to display a window you define by picking a center point and a magnification value or height. A smaller value for the height increases the magnification. A larger value decreases the magnification.

Prompts

1 Specify center point: **pick a point**

2 Enter magnification or height <226.66>: **enter a value**

Menu Location: View, Zoom>
Prerequisite: None
Keyboard Command: ZOOM, C

Zoom Extents

This command zooms to display the drawing extents. You can use Zoom Extents transparently, but it always regenerates the drawing.

Menu Location: View, Zoom>
Prerequisite: None
Keyboard Command: ZOOM, E

Zoom IN

This command increases the zoom factor of the current viewport by a factor of 2.0.

Menu Location: View, Zoom>
Prerequisite: None
Keyboard Command: ZOOM, 2.0x

Zoom OUT

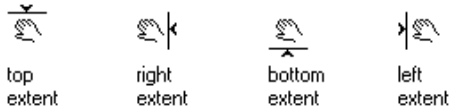
This command decreases the zoom factor of the current viewport by a factor of 0.5.

Menu Location: View, Zoom>
Prerequisite: None
Keyboard Command: ZOOM, 0.5x

Pan

This command moves the drawing display in the current viewport. The cursor changes to a hand cursor. By holding down the pick button on the pointing device, you lock the cursor to its current location relative to the viewport coordinate system. The drawing display is moved in the same direction as the cursor.

When you reach a logical extent (the edge of the drawing space), a bar is displayed on the hand cursor on the side where the extent has been reached. Depending on whether the logical extent is at the top, bottom, or side of the drawing, the bar is either horizontal (top or bottom) or vertical (left or right side).



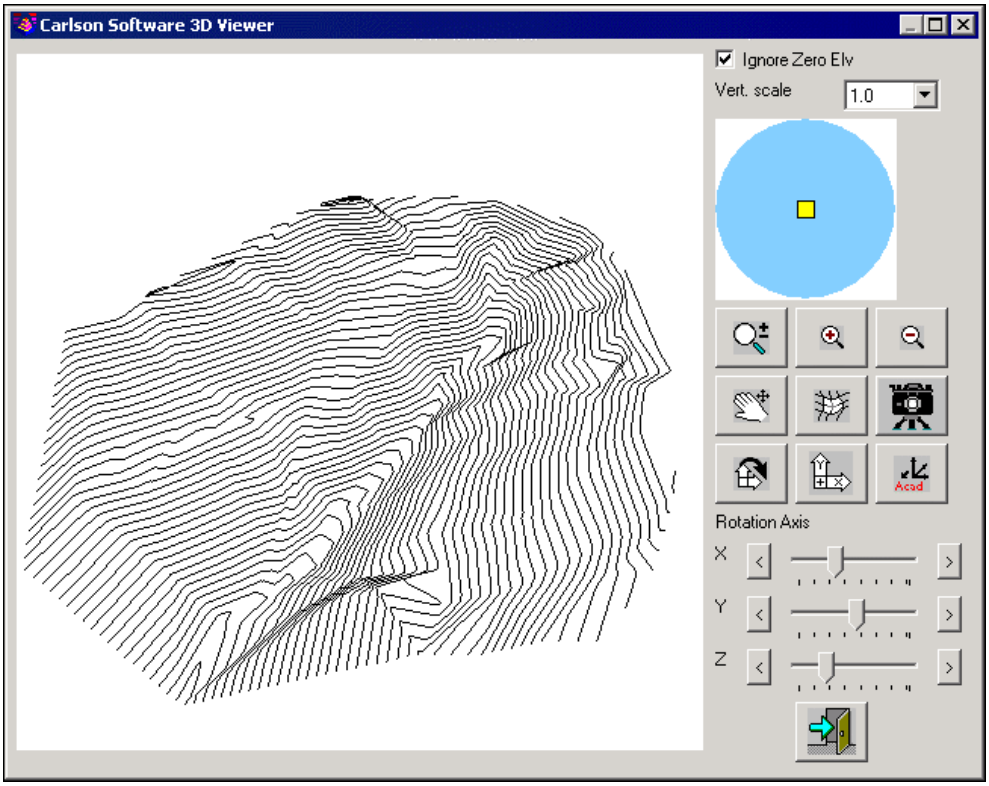
When you release the pick button, panning stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again to pan the display from that location.

To stop panning at any time, press Enter or ESC.

Menu Location: View
Prerequisite: None
Keyboard Command: P

3D Viewer Window

This command allows you to view the selected 3D faces, polylines, lines, and points in 3D using the OpenGL graphics library for rendering, which gives it superior performance. This command gives you the ability to zoom in and out, pan, rotate around the X, Y, or Z axis, and shade in user-positioned lighting. You also have the option to export the view to a windows bitmap (.BMP) file. The window below is displayed after you select the entities to be shown. To pan the display, click on the image and drag the pointer.



- **Ignore Zero Elv**: This option excludes entities on zero elevation from the display.
- **Vert**. **Scale**: Controls the vertical exaggeration of the 3D display.
- **Light Position**: This control represents the position of the sun in the sky looking from above. The position of the sun in the center of the control means the sun is in a zenith, and a position near the edge of the circle means the sun is near the horizon. To move the sun, simply drag it to new location or click there.
- **Dynamic Zoom Mode**: Allows you to zoom in and out by holding the left mouse button down and moving the mouse up and down.
- **Zoom In**: This option incrementally zooms in to the view.
- **Zoom Out**: This option incrementally zooms out to the view.
- **Pan Mode**: Allows you to pan the display
- **Shade Model**: This option allows you to shade the surfaces of your view. The color of the shade depends on the lighting of the surface, which you adjust using the Light Position control in the middle of the controls window. See Light Position below.
- **Export to BMP**: This option creates a .BMP bitmap file from your current view.
- **Rotation Mode**: Allows you to rotate the view in the X or Y axis while the cursor is in the middle of the display. Rotates on the Z axis when the cursor is near any edge of the display. The cursor will indicate the axis.
- **Plan View**: This option restores the X, Y, and Z axes back to zero.
- **Set Acad View**: This option allows you to send the current 3D view back to the AgStar screen when you exit the 3D Viewer window. You update the current CAD view to match the view that is displayed in the 3D viewer window.
- **Z-Axis**: This slider rotates the view on the Z axis. The range of these sliders is -180 to +180 degrees with middle being 0, which is the default position when the viewer starts.
- **Y-Axis**: This slider rotates the view on the Y axis.
- **X-Axis**: This slider rotates the view on the X axis.

Menu Location: View
Prerequisite: Entities to display.
Keyboard Command: CUBE

Regen

This command regenerates the drawing and refreshes the current viewport.

Menu Location: View
Prerequisite: None
Keyboard Command: REGEN

Twist Screen Standard

This command allows you to "twist" the screen's orientation so that a direction other than North is toward the top of the screen and the drawing. It does not do a coordinate rotation, and it leaves the database unchanged. The ROTATE and MOVE commands in the Edit menu can be used to do a coordinate rotation and translation.

This command prompts you for the twist angle, then adjusts the screen and crosshairs to that angle. The twist angle is always measured counterclockwise, with 0 degrees at the east/right.

Menu Location: View, Twist Screen>
Prerequisite: None
Keyboard Command: TWIST1

Twist Screen Line

This command is a variation of Twist Screen Standard. The command aligns a selected line to be parallel to the east-west direction of your graphics screen.

Think of the line you select as a pointer or arrow that will be moved to point in the east direction of the screen. Select the line closest to the line endpoint that you want to point in the horizontal or east direction of the screen.

Menu Location: View, Twist Screen>
Prerequisite: None
Keyboard Command: TWIST2

Twist Screen Surveyor

This command is another variation of Twist Screen Standard. You enter the angle/azimuth that you want to be aligned parallel to the east-west direction of the graphics screen.

Menu Location: View, Twist Screen>
Prerequisite: None
Keyboard Command: TWIST3

Restore Due North

This command twists the screen to make due north vertical.

Menu Location: View, Twist Screen>
Prerequisite: None
Keyboard Command: TWIST4

Layer ID

This command reports the layer name of the selected entity.

Prompts

Pick entity to read layer: *pick an entity*

Layer: FINAL

Pick entity to read layer: *Press Enter to end*

Pull-Down Menu Location: View
Prerequisite: None.
Keyboard Command: layerid

Change Layer

This command allows you to change the layer of a group of entities you select. The layer name can be either typed or read from an existing entity you select.

Prompts

1 Select entities to be changed.

Select objects: **pick entities**

2 Pick entity with new layer or press Enter to type name: **pick another entity**

The first selection is now assigned to the layer of the second selection.

Menu Location: View
Prerequisite: Entities to change.
Keyboard Command: LCHG

Freeze Layer

This command allows you to freeze layers assigned to the entities you select.

Menu Location: View
Prerequisite: None
Keyboard Command: LOFF

Isolate Layer

This command allows you to freeze all layers except the layers assigned to the entities you select. The program prompts to see if you would like to retain the PNT layers, which keeps the point layers from freezing. These layers include PNT, PNTMARK, PNTELEV, PNTDESC, and PNTELEV.

Prompts

1 Select objects on layers to isolate.

Select objects: **pick entities**

2 Retain POINT layers [Yes/<No>]? **Press Enter**

Menu Location: View
Prerequisite: None
Keyboard Command: ISOLATE

Restore Layer

This command thaws the layers frozen by the Isolate Layer command.

Menu Location: View
Prerequisite: You must have previously run the Isolate Layer command
Keyboard Command: RESTORE

Thaw Layer

This command allows you to thaw the layers frozen by the Freeze Layer command.

Menu Location: View
Prerequisite: None
Keyboard Command: LON

List

Displays database information for selected objects.

Carlson Survey lists the object type, object layer, and X,Y,Z position relative to the current user coordinate system (UCS) and whether the object is in model space or paper space.

LIST reports color, linetype, and lineweight information if these items are not set to BYLAYER. The thickness of an object is displayed if it is nonzero. Z coordinate information defines the elevation. If the extrusion direction of the entry differs from the Z axis (0,0,1) of the current UCS, LIST also reports the extrusion direction in UCS coordinates.

LIST reports additional information related to the specific object selected.

Menu Location: View
Prerequisite: None
Keyboard Command: LIST

Polyline Info

This command reports the length and elevation of the selected line or polyline.

Prompts

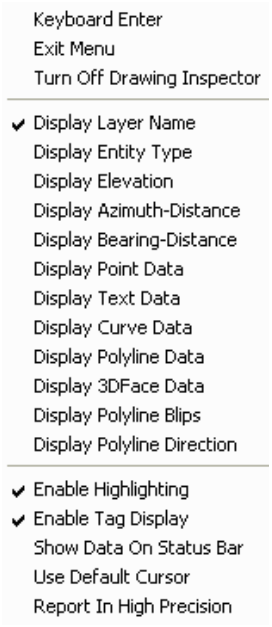
Pick Polyline or Line: *pick a polyline*

Polyline length: 145.43 Elevation: 100.0

Pull-Down Menu Location: View
Keyboard Command: polylen
Prerequisite: A polyline

Drawing Inspector

This command reports object properties to you as you move the cursor over an entity. You can simply move the pointer over an entity and the selected property will be displayed either in a pop-up window next to the pointer and/or on the status bar, depending on the selected option. Drawing Inspector is a transparent command that can run while other commands are running. Once Drawing Inspector is started, it will stay active even while running other commands until you turn it off. To turn off Drawing Inspector, run the command again to toggle it off by pick Drawing Inspector from the Inquiry pull-down menu or from the toolbar or by typing the command name, or right-click and choose Turn off Drawing Inspector. The options for this command are set in the menu that pops up by clicking the right mouse button. The available properties are: Layer Name, Elevation, Azimuth-Distance, Bearing-Distance, Point Data, Text Data, Curve Data, 3D Face Data, Polyline Data and Polyline Blips.



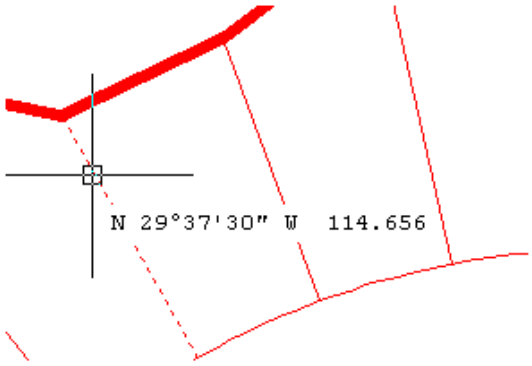
In the *Drawing Inspector* menu, you can choose one or more properties to display.

Display Layer Name: Allows you to display the layer name of the entity.
Display Entity Type: Allows you to display the type of the entity (ie. TEXT or POLYLINE).
Display Elevation: Allows you to display the elevation of the entity.

Display Azimuth-Distance: Allows you to display the azimuth and distance of a line.
Display Bearing-Distance: Allows you to display the bearing and distance of a line.
Display Point Data: Allows you to display the coordinate data of point.
Display Text Data: Allows you to display the attributes of text.
Display Curve Data: Allows you to display the radius, arc length, chord length and delta angle of a curve.
Display Polyline Data: Allows you to display the end point elevations, horizontal distance, slope distance and slope ratios.
Display 3D Face Data: Allows you to display the Z elevations at the face corners.
Display Polyline Blips: Allows you to displays temporary blip plus marks at the vertice locations of polylines.
Display Polyline Direction: Allows you to displays temporary arrows to show the direction of polylines.

In the *Drawing Inspector* menu, you can also choose how the property information is reported.

Enable Highlighting: Allows you to highlight the object that the *Drawing Inspector* is reporting.
Enable Tag Display: Enables you to view the information next to the cursor on the screen.
Show Data On Status Bar: Enables you to view the information on the status bar, in the lower corner of the screen.
Use Default Cursor: When enabled, only the drawing cursor shows. When disabled, the mouse pointer is also shown.
Report In High Precision: When enabled, displays 8 decimals on distance and 4 decimal seconds on angles.



Example of Drawing Inspector reporting
Bearing-Distance using the Tag Display

Pulldown Menu Location: View
Keyboard Command: inspector
Prerequisite: None

Draw Commands 5

Line

This command allows you to draw a line entity by picking points on the screen or by supplying the coordinate values using the point number and associated coordinates stored in the current coordinate file. The Line command links the line with the points when the line is drawn using point numbers if the Link Linework with Points option is turned on. This option is set under General Settings in the Configure command in the Settings menu. With links active, changing a point with a command like Move Points automatically updates the line. This command always draws 2D lines with a zero elevation.

Prompts

1 Pick point or point numbers: **1-3**

You may enter a single point number or a range of point numbers

2 Undo/Distance/<Pick point or point numbers>: **16**

3 Undo/+/-/Close/Distance/<Pick point or point numbers>: **35**

4 Undo/+/-/Close/Distance/<Pick point or point numbers>: **+**

The + or - activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line.

5 Perpendicular Distance Right: **80**

6 Undo/+/-/Close/Distance/<Pick point or point numbers>: **-**

The + or - activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line.

7 Perpendicular Distance Left: **105.12**

8 Undo/+/-/Close/Distance/<Pick point or point numbers>: **D**

The distance option allows you to input a distance for the next line segment. The position of the cursor determines the angle.

9 Enter distance: **174.32**

10Undo/+/-/Close/Distance/<Pick point or point numbers>: **C**

The close option draws a line segment back to the original starting point

Menu Location: Draw
Prerequisite: None
Keyboard Command: 2DLINE

2D Polyline

Function

This command draws a polyline entity. You can either provide point numbers from the current coordinate file or pick points on the screen to define coordinates. A Polyline is a series of line and/or arc segments joined together in one entity or object.

Prompts

1 Pick point or point numbers: **pick point**

2 Undo/Arc/Length/<Pick point or point numbers>: **3**

3 Undo/+/-/Arc/Close/Length/<Pick point or point numbers>: **+**

The + or - activates an additional prompt option that allows you to plot line segments at a 90 degree deflection angle from the last line. This is useful for plotting buildings. See also the Extend by Distance command on the Edit menu.

4 Perpendicular Distance Right: **50**

5 Arc/Length/+/-/Close/Undo/<Pick To point or point number>: **C**

This closes the polyline and ends the command.

Menu Location: Draw

Prerequisite: None

Keyboard Command: 2DP

3D Polyline

Function

This command allows you to set a specified layer and plot a 3D polyline which can be used to define breaklines and features to better define surface models. You can pick points from the graphics screen or use point numbers from the current coordinate file. You can also specify a range of point numbers. For example, entering "3-1,5,10" would draw 3D Line segments from point number 3 to 2 to 1 to 5 to 10.

You can choose to pick points from a surface model that is stored in a file, which can be either a grid (.GRD) file or triangulation (.FLT) file. When a point is picked or a point number is entered, the program uses the northing and easting of this point and calculates the elevation from the surface model.

When you create a 3D polyline by picking points from the screen, be careful not to miss a point resulting in a zero elevation. This creates a 3D polyline that drops to zero at one point. As points are picked, Draw 3D Polyline reports the coordinate so you can check that the correct elevation was used. If you miss a point and see a zero elevation, you can type Undo at the next prompt.

Prompts

1 Layer Name for 3DPoly <BREAKLINE>: **Press Enter**

2 Prompt for elevations (.XY filter) (Yes/<No>)? **Y**

Using the .XY filter allows you to pick the X and Y coordinate from the screen and type in the elevation. If you enter N, then the Z coordinate of the point picked will be applied.

3 Use surface model from file (Yes/<No>)? **No**

If you choose Yes, then when a point is picked the elevation of the picked point is calculated using a surface model. The surface model can be either a grid (.GRD) file or triangulation (.FLT) file.

4 Undo/<Pick point or point numbers>: **pick point**

5 Elevation <0.0>: **99.5**

6 Undo/<Pick point or point numbers>: **15**

This is a point number from the current coordinate file.

7 Undo/<Pick point or point numbers>: **pick point**

8 Percent slope/Ratio slope/Elevation <99.5>: **P** for Percent slope.

9 Ratio slope/Elevation/Percent slope <-2.53>: **-2**

Sets the elevation of this point to make a 2 percent slope from the previous point.

10Undo/Close/<Pick point or point numbers>: **Press Enter**

Pressing Enter ends the command.

11Draw another 3D polyline [Yes/<No>]? **N**

Menu Location: Draw

Prerequisite: None

Keyboard Command: 3DPLINE

Circle

This command allows you to draw a circle.

Prompts

1 Pick center point or point number or [3P/2P/TTR]: **pick point or specify option**

- **3P**: This option draws a circle based on three points on the circumference.
- **2P**: This option draws a circle based on two endpoints of the diameter.
- **TTR-Tangent, Tangent, Radius**: This option draws a circle with a specified radius tangent to two objects.

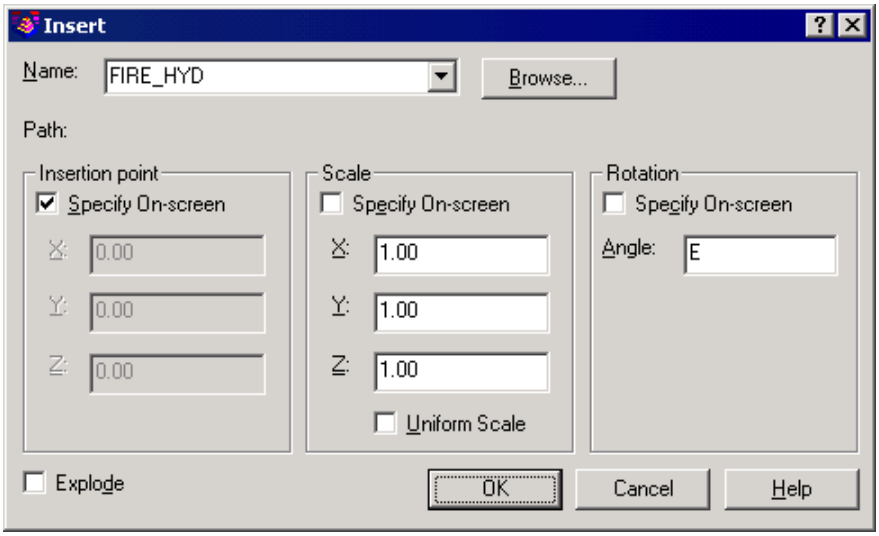
2 Specify radius of circle or [Diameter]: **enter a value**

Sometimes more than one circle matches the criteria specified in the command. The circle whose tangent points are closest to the selected points is drawn.

Menu Location: Draw
Prerequisite: None
Keyboard Command: SCIRCLE

Insert

This command allows you to place a named block or drawing into the current drawing.



1 In the Insert dialog box, you specify the block to insert and define the position for the inserted block. The last block you insert during the current editing session becomes the default block for subsequent uses of this command.

- **Name**: This field specifies the name of a block to insert or the name of a file to insert as a block.
- **Browse**: This button opens the Select Drawing File dialog box (a standard file selection dialog box) where you can select a block or a file to insert.

2 Under Insertion Point, you specify the insertion point for the block.

- **Specify On-Screen**: This option specifies the insertion point of the block using the pointing device.
- **X**: This field sets the X coordinate value.
- **Y**: This field sets the Y coordinate value.

- **Z**: This field sets the Z coordinate value.

3 Under Scale, you specify the scale for the inserted block. Specifying negative values for the X, Y, and Z scale factors inserts a mirror image of a block.

- **Specify On-Screen**: This option specifies the insertion point of the block using the pointing device.
- **X**: This field sets the X coordinate value.
- **Y**: This field sets the Y coordinate value.
- **Z**: This field sets the Z coordinate value.
- **Uniform Scale**: This option specifies a single scale value for X, Y, and Z coordinates. A value specified for X is also reflected in the Y and Z values.

4 Under Rotation, you specify the rotation angle for the inserted block.

- **Specify On-Screen**: This option specifies the rotation angle of the block using the pointing device.
- **Angle**: This field sets a rotation angle for the inserted block.

5 You can explode the block and inserts to the individual parts of the block. When you select Explode, you specify only an X scale factor.

Menu Location: Draw
Prerequisite: None
Keyboard Command: DDINSERT

Text

Creates a single-line text object.

You can use the TEXT to enter several lines of text that you can rotate, justify, and resize. As you type at the Enter Text prompt, the text you are typing is displayed on the screen. Each line of text is a separate object. To end a line and begin another, press Enter after entering characters at the Enter Text prompt. To end the TEXT command, press Enter without entering any characters at the Enter Text prompt.

By applying a style to the text, you can use a variety of character patterns or fonts that you can stretch, compress, make oblique, mirror, or align in a vertical column.

If TEXT was the last command entered, pressing ENTER at the Specify Start Point of Text prompt skips the prompts for height and rotation angle and immediately displays the Enter Text prompt. The text is placed directly beneath the previous line of text. The point specified at the prompt is also stored as the Insertion Point object snap.

Prompts

1 Current text style: "MONO" Text height: 4.00

2 Specify start point of text or [Justify/Style]: **S**

The style option lets you change the textstyle on the fly

3 Enter style name or [?] <MONO>: **STANDARD**

4 Current text style: "STANDARD" Text height: 4.00

5 Specify start point of text or [Justify/Style]: **J**

The justify option lets you specify the justification for the text.

6 Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]: **BC**

In this case BC = Bottom Center

7 Specify bottom-center point of text: **pick point or enter coordinates**

8 Specify height <4.00>: press enter to select default or enter text height

9 Specify rotation angle of text <0d0'0">: press enter to select default or enter angle

10Enter text: Found Iron Pin

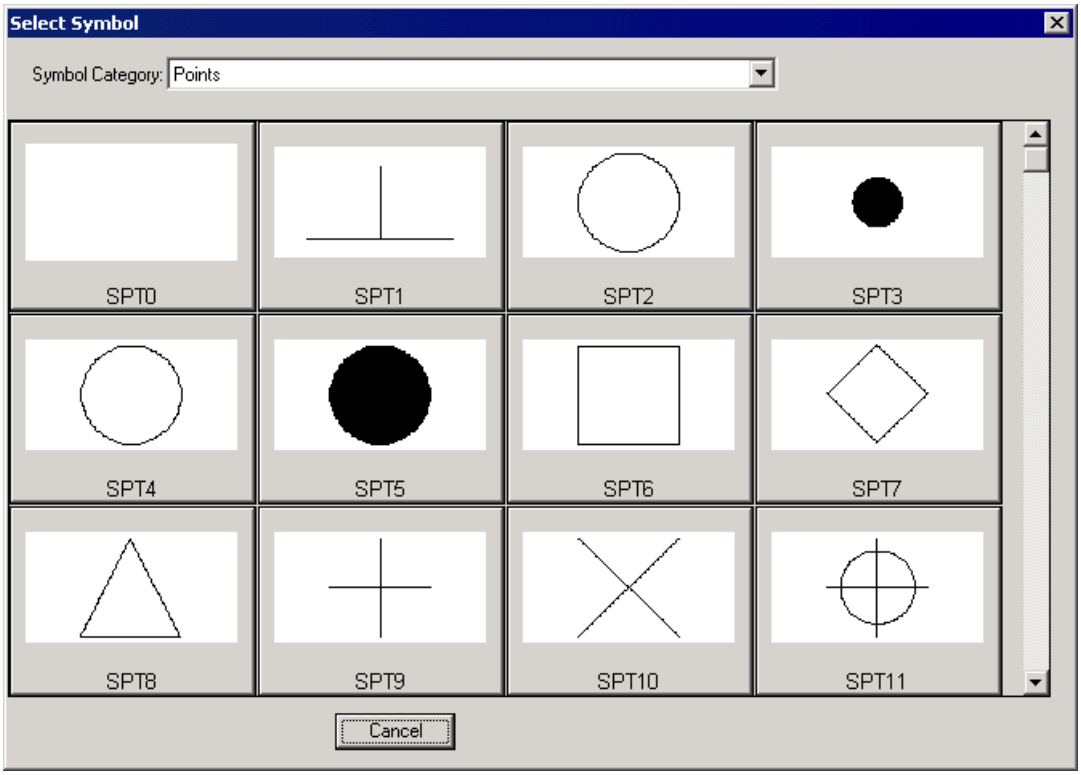
11Enter text: press enter to end

Menu Location: Draw
Prerequisite: None
Keyboard Command: DTEXT, TEXT

Insert Symbols

This command inserts symbols from the symbol library into the drawing. The symbol library may be edited using the Edit Symbol Library command. The locations for the symbols can be specified by picking points, specifying point numbers in the current coordinate (.CRD) file or by entering the northing and easting. If you specify a point number and that point number already has a symbol on it, you will be prompted whether or not to replace the existing symbol. Selecting the Enter coords option allows you to insert the symbol by entering a northing and easting. Using the Select entities option, symbols can also be placed on arcs, points, lines or polylines. Under the Options command, you can turn prompting for rotation on or off. With rotation off, the symbol will be inserted horizontal to the current twist screen.

Choose a symbol from the Select Symbol dialog by clicking on it. You may select a different category by choosing the Symbol Category drop down list. Within each category, use the scroll bar to view all of the symbols.



Prompts

1 Options/Select entities/Enter coords/<Point numbers or pick point>: **pick point**

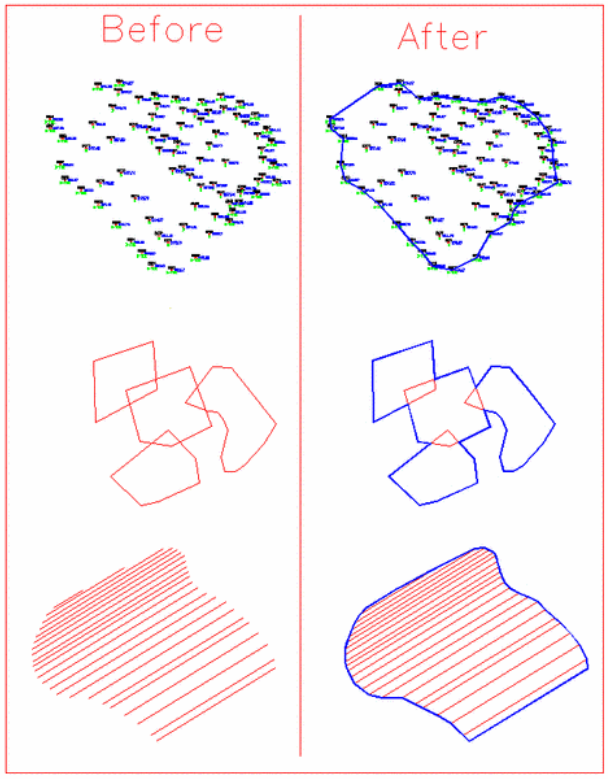
2 Options/Select entities/Enter coords/<Point numbers or pick point>: **5-10**

3 Options/Select entities/Enter coords/<Point numbers or pick point>: **Press Enter**

Menu Location: Draw
Prerequisite: None
Keyboard Command: PTSYM

Shrink-Wrap Entities

This command creates a closed polyline which encloses a given set of entities. The resulting polyline is created in the current layer. The program works on either point entities or polylines. For points, the program creates a closed polyline through the points around the perimeter of the area defined by the points. For polylines, the shrink-wrap polyline follows the outside border of the selected polylines. The polylines that are processed have to be connected to be shrink-wrapped. The snap tolerance is the maximum gap that will be joined to make the closed polyline. For open polylines, as in the bottom figure, the Gap method works better, as it jumps across the gaps and connects the end points.



Prompts

Shrink-wrap across gaps or bounded linework only [<Gap>/Bound]? *G*
Shrink-wrap layer <FINAL>:
Select points and linework to shrink-wrap.
Select objects: *select entities to process*
Reading points... 46
Inserted 46 points.
Inserted 23 breakline segments
Perimeter reduction level 0-3 (0-None, 3-Most) <2>: *2*
Reduce Perimeter Pass: 1 Removed: 5
Reduce Perimeter Pass: 2 Removed: 3
Reduce Perimeter Pass: 3 Removed: 4
Reduce Perimeter Pass: 4 Removed: 2
Reduce Perimeter Pass: 5 Removed: 1
Reduce Perimeter Pass: 6 Removed: 0
Create 2D or 3D Polyline [<2D>/3D]? *2D*

Pulldown Menu Location: Draw
Keyboard Command: swplines
Prerequisite: Entities

3 Point Curve

This command draws an arc between three points. The first point is the PC, the second is a point on the arc, and the third is the PT. The points can either be picked on-screen or specified by point number.

Prompts

1 Pick PC point or point numbers: **101**

2 Pick Second point or point number: **102**

3 Pick PT point or point number: **103**

Menu Location: Draw, Curves >
Prerequisite: None
Keyboard Command: 3PA

PC PT Radius Point

This command draws an arc between the PC point, radius point, and PT point. The points can either be picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counterclockwise. The program shows one direction and asks if it is correct. If you want the arc to go in the other direction, enter No.

Prompts

1 Pick PC point or point number: **101**

2 Pick Radius point or point number: **102**

3 Pick PT point or point number: **103**

4 Is the direction of this arc correct ? No/<Yes>: **N**

The arc draws in the other direction.

Menu Location: Draw, Curves >
Prerequisite: None
Keyboard Command: PCA

PC Radius Chord

This command draws an arc given the PC point, radius length, chord length, and chord bearing. The PC point can either be picked on-screen or specified by point number. Given these points, the arc can be drawn clockwise or counterclockwise. The program shows one direction and asks if it is correct. If you want the arc to go in the other direction, enter No.

Prompts

1 Radius of Arc <-40.00>: **500**

2 PC Start Point ?

Pick point or point number: **pick a point**

3 Chord bearing or chord endpoint (<Bearing>/Point)? **Press Enter**

4 Enter Bearing (Qdd.mmss) <90.0000>: **145.1041** (for NE 45°10'41")

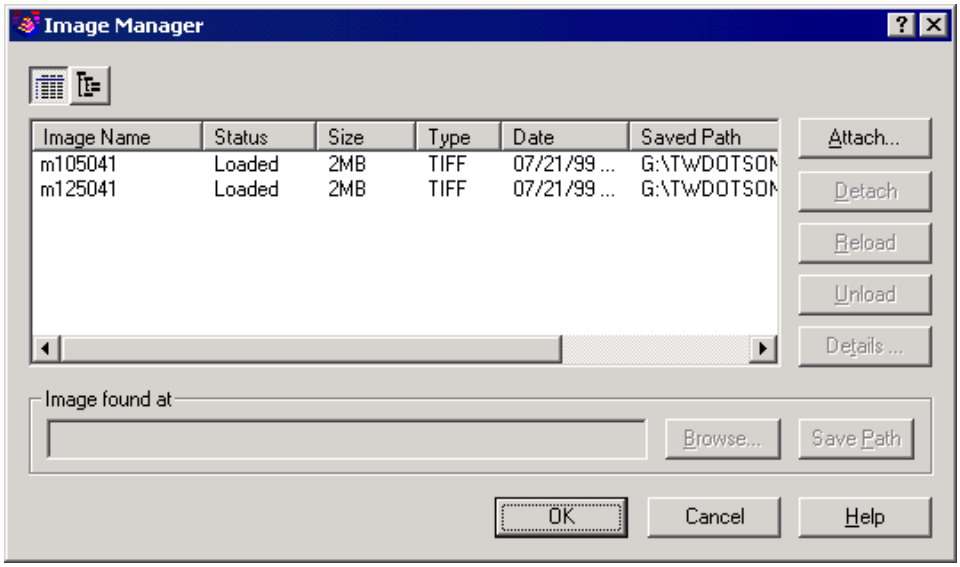
5 Chord Length <200.46>: **200**

6 Is this arc in the correct direction (<Yes>/No)? **Press Enter**

Menu Location: Draw, Curves >
Prerequisite: None
Keyboard Command: SRCB

Raster Image

This command allows you to manage raster images.



1 The Image Manager dialog box lists all the image files attached to the current drawing. You can view the parameters and details for selected images. You can attach new image files and detach, locate, reload, and unload existing images.

- **List View**: This button lists the image definitions attached to the drawing. Each image name appears only once regardless of how many times you attach (insert) the image. You can sort the list of images by name, status (loaded, unloaded, or not found), size, type (TIFF, for example), date, or the saved path and file name. By default, Carlson Survey displays the list alphabetically by image name.

To select multiple images, hold down SHIFT or CTRL while selecting items.

To sort the list alphabetically or numerically by a specific column, click that column's heading.

To change the width of the column, drag the line between the column headings to the right or left. The program saves and restores the settings when you reopen the dialog box.

To change an image name, select it and then click it again, or select it and then press F2. You cannot edit names of images that reside in external references (xrefs). Image names can include up to 255 characters and can contain letters, digits, spaces, and any special characters not used by Microsoft® Windows® or Carlson Survey. The image name can be identical to the file name, but changing the image name does not change the file name.

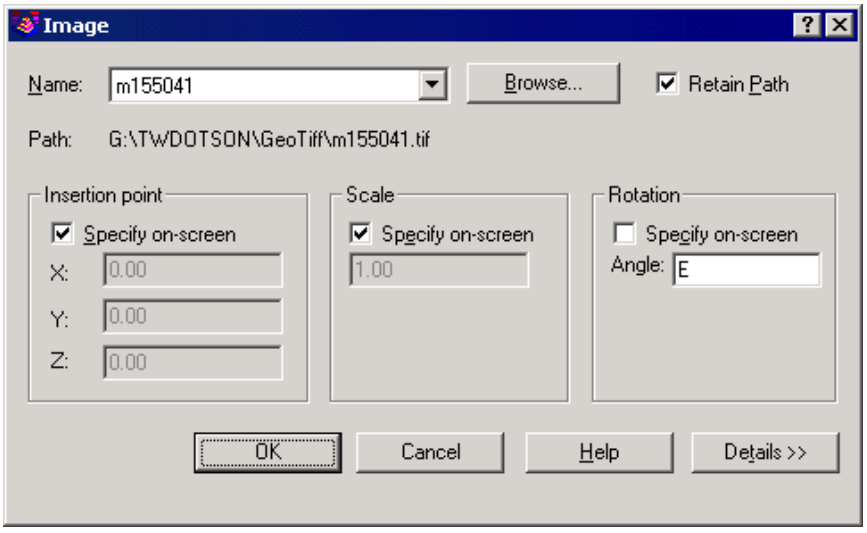
- **Tree View**: This button displays all the image definitions and the levels of nesting of images within xrefs. The top level of the tree view shows images that you attached directly to the drawing, images nested in block references, and the names of externally referenced drawings containing images. The names of the images attached to the externally referenced drawings appear nested within the drawing at the next tree level. To insert a copy of an already attached image, select it, and then choose Attach.

Tree view lists the image names only (not file names) and lists the image name just once, regardless of how many times you attach (insert) the image.

You can edit an image name by selecting it and then clicking it again, or by selecting it and then pressing F2. However, you cannot select more than one image at a time.

- **Attach**: This option displays the Select Image File dialog box. When you unload and then reload an image, the program draws that image on top. Images remain loaded or unloaded from one drawing session to the next.
- **Detach**: This option removes the selected image definitions from the drawing database and erases all the associated image objects from the drawing and from the display.
- **Reload**: This option loads the most recent version of an image or reloads an image that was previously unloaded. Reloading does not control whether the image is displayed, but it ensures display of the most current image.
- **Unload**: This option unloads image data from working memory without erasing the image objects from the drawing. It is recommended that you unload images no longer needed for editing to improve performance. An unloaded image cannot be displayed or plotted. You can selectively load and unload individual images from a working list of images associated with the drawing file.
- **Details**: This option opens the Image File Details dialog box, which displays the image name, saved path, active path, file creation date and time, file size and type, color system, color depth, width and height in pixels, resolution, default size in units, and a preview image.
- **Image Found At**: This field shows the path of the selected image. If you select multiple images, this field remains blank. The path shown is the actual path where the image resides.
- **Browse**: This option opens the Select Image File dialog box (a standard file selection dialog box). The path you select appears under Image Found At.
- **Save Path**: This option stores the new path information. Press ESC while editing the path to restore the old path. If the program cannot find the referenced image in the new path, the image's status changes to Not Found. If you do not choose Save Path after editing the path, the program uses the original image path the next time you load the drawing.

2 Under the Image dialog box, you can attach an image.



3 In the Image dialog box, you must first identify the image and the path.

- **Name**: This field identifies the image you have selected to attach, either from the Select Image File dialog box (an unattached image) or from the list of previously attached images. To add another instance of an image file that is already attached, select the image name from the list and choose OK.
- **Browse**: This option opens the Select Image File dialog box (a standard file selection dialog box). If Show Preview is selected, the program displays a preview of the selected file.
- **Retain Path**: This option saves the path of the image file with the image definition. If Retain Path is not selected, only the image name is saved and AgStar searches the Support File Search Path.

4 Under Insertion Point, you must specify the insertion point for the selected image. Specify On-Screen is the default. The default insertion point is 0,0.

- **Specify On-Screen**: This option directs input to the command line or the pointing device. If Specify On-Screen is cleared, enter the insertion point in X, Y, and Z.
- **X**: This field sets the X coordinate value.
- **Y**: This field sets the Y coordinate value.
- **Z**: This field sets the Z coordinate value.

5 Under Scale, you must specify the scale factor of the selected image. Specify On-Screen directs input to the command line or the pointing device. If Specify On-Screen is cleared, enter a value for the scale factor. The default scale factor is 1.

6 Under Rotation, you must specify the rotation angle of the selected image. If Specify On-Screen is selected, you may wait until you exit the dialog box to rotate the object with your pointing device or enter a rotation angle value on the command line. If Specify On-Screen is cleared, enter the rotation angle value in the dialog box. The default rotation angle is 0.

Menu Location: Draw
Prerequisite: Raster image
Keyboard Command: IMAGE

Place Image by World File

This function allows you to insert Geo-Referenced image files into AutoCAD drawings. This process requires the presence of an accompanying World file. The TFW file contains information about the location and scaling of the actual raster image TIF file. This eliminates the guesswork in inserting, moving, and rotating raster images to the project area. You begin by selecting the TFW or JGW file to process. If the related TIF file is present in the same directory, the image will be inserted into the proper coordinates.

Prompts

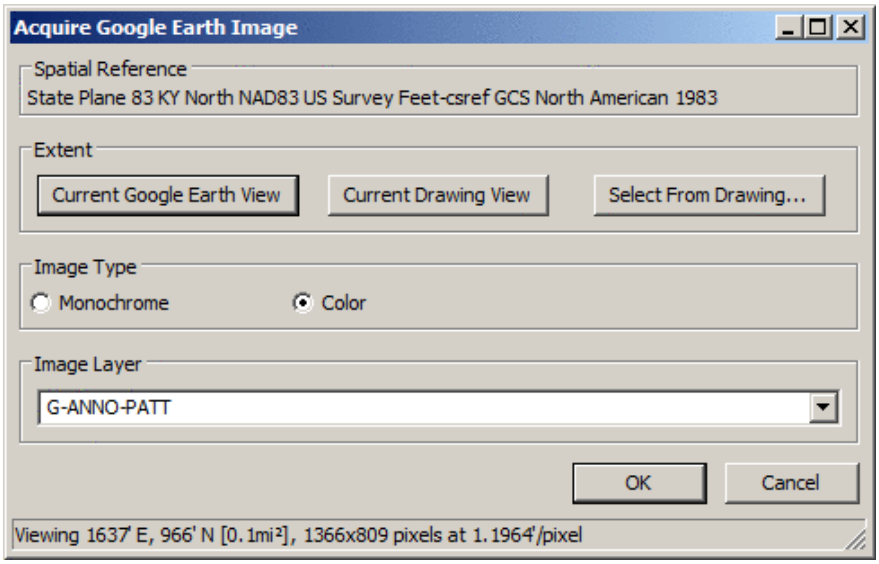
Select World File: *choose existing .TFW or .JGW file*

Pulldown Menu Location: Draw > Images
Keyboard Command: geotiff
Prerequisite: A georeferenced image file

Place Google Earth Image

In addition to providing a graphical method for displaying feature-rich data located anywhere on the globe, Google Earth also provides the ability for software applications to extract its aerial imagery. While the positional accuracy of the Google Earth surface should be considered "approximate," it might be suitable for preliminary land-planning studies or "proof-of-concept" preliminary designs.

Consider the following example. Based on the physical screen size of the Google Earth application and the "zoom" (or "view") resolution of a project site, the following values (summarized at the bottom of the dialog box) were returned:



Unit	Horizontal	Vertical
Feet	1637	966
Pixels	1366	809
Feet/Pixel	1.19	1.19

Google Earth View Area

In the sample above, the total area is calculated and displayed (0.1 mi^2) along with the desired "spatial reference" coordinate system for our project site.

Spatial Reference: Displays the spatial reference coordinate projection system of the current drawing. The projection can be set using the Drawing Setup command.

Extent - Current Google Earth View: Gets the overall dimensions of the Google Earth session and displays the results in both pixels and the appropriate units of measure.

Extent - Current Drawing View: Gets the overall dimensions of the current CAD view and displays the results in both pixels and the appropriate units of measure.

Extent - Select from Drawing: Sets the overall dimensions of the Google Earth session to conform with a drawing window from CAD and displays the results in both pixels and the appropriate units of measure.

Image Type: Allows the ability to indicate the type of image to be placed into the drawing. Gray-scale images can be inserted into the drawing automatically, color drawings must be manually saved from Google Earth and then selected for insertion in the CAD drawing (see the *Prompts* section below).

Image Layer: Allows the ability to indicate the layer upon which the image should be inserted.

Note:

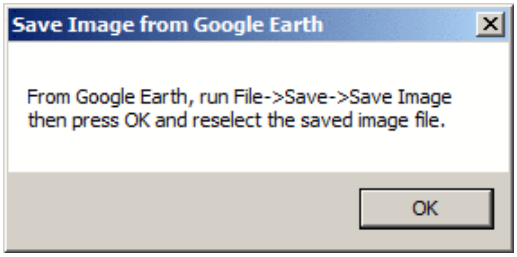
- Once an image has been inserted into the CAD drawing, it may be helpful to create an associated "world" file for the image in case it needs to be re-inserted into the active drawing (or an alternate drawing). Creating a "world" file assists with this task. To generate a world file, please reference the Create by Image in Drawing command.
- Once the image has been inserted into the CAD drawing, it may be helpful to control its "display order" by using the View > Display Order > Order by Layer command or View > Display Order > Send to Back command.
- The Place Google Earth Image routine fetches aerial imagery in real-time from the Google servers and requires an Internet connection to proceed. In the event that an Internet connection is not available, the following error message may be displayed: "*Failed to initialize Google Earth. Please ensure Google Earth client software is functional and online*"

- It bears repeating that the aerial imagery returned by Google Earth should only be used for illustrative or proof-of-concept purposes only!

Prompts

First corner/Identify Entity <Current view>: *Identify one corner of a drawing window that should be used to set the Google Earth display or pick an existing entity with discreet upper and lower bounds to define a region or press Enter to use the current view (this is the same as the "Extent - Current Drawing View" option above).*

Specify opposite corner: *Identify the opposite corner of a drawing window that should be used to set the Google Earth display.*

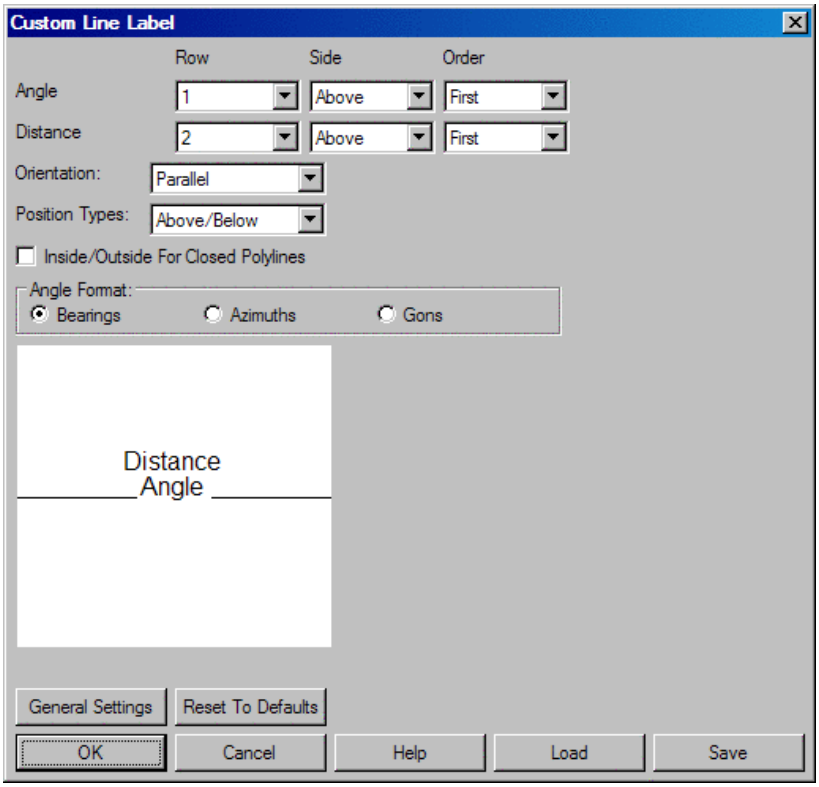


For Color images: *Use Windows functionality (Alt+Tab) to switch focus to the Google Earth application and use the Google Earth > File > Save > Save Image command (Ctrl+Alt+S) to save the current Google Earth image.* Once saved,
Click OK on the Carlson alert dialog box shown above and locate/select the image just saved. Click Open when the file has been located

Pulldown Menu Location: Draw > Images
Keyboard Command: google_image
Prerequisite: Coordinate projection system, Functioning version of Google Earth, Internet connection

Custom Linework Label Formatter

This command allows you to customize the labeling for lines and polylines. You are first prompted to select a line or polyline to label, given the existing defaults currently set. The linework is shown as labeled on the screen. The command line, shown below, also offers you an important choice called Options. When you type 'O' for options the below dialog box appears. In this dialog, there are three columns at the top of the dialog, along with other features. On the command line, there is also a choice called Format (F), which allows you to enter quick-key style keywords for quickly changing the label format. See below for these



Row: This column allows you to stack the data in different ways. You can place more than one item in the same row. If *None* is selected, then that item will not be displayed.
Side: This column allows you to place each item either inside or outside of the line or polyline.
Order: This column determines the order of items when they are placed in the same row.
General Settings: This button brings you to the Annotate Defaults dialog, see 'Annotate Defaults' for more.
Reset To Defaults: This button restores the default settings shown above.
Load/Save: You may also Load and Save different label configurations with the corresponding buttons.

Prompts

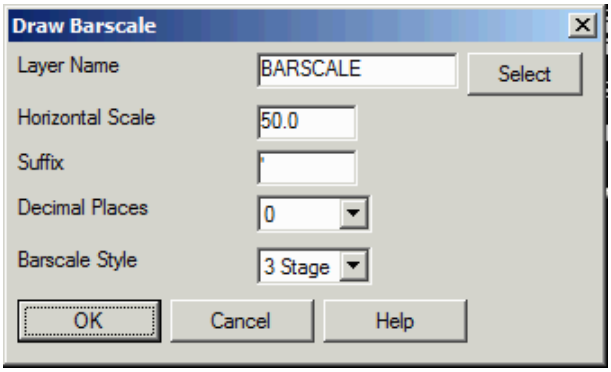
Options/Format/Points/<Select line or polyline>: *select entity*
Options/Format/Points/<Select line or polyline>: O
Custom Line Label dialog *choose your preferences and click OK*
You can decide to go into the Option dialog at the start of the command, or after your initial labeling. If you use the Format command line option, you will be asked to enter the Format command. The choices are:

B = bearing
A = azimuth
G = gon
D = distance
R = next row
_ = switch side of line

Pulldown Menu Location: Draw
Keyboard Command: annline
Prerequisite: An arc to label

Draw Barscale

This command draws a barscale at the user-specified scale. The command options are set in the dialog shown here. The Horizontal Scale controls the size and labels for the barscale. For example, enter 50 for 1 inch = 50 feet in English mode. The Barscale Style chooses between different barscale formats.



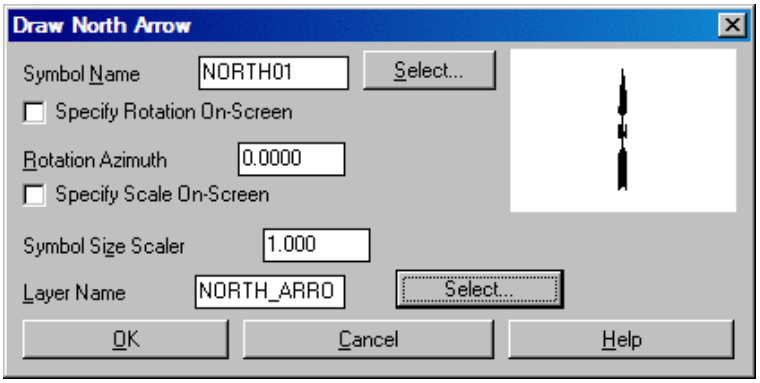
Prompts

Draw Barscale options dialog
Pick location for barscale: *pick a point*

Pulldown Menu Location: Draw
Keyboard Command: barscale
Prerequisite: None

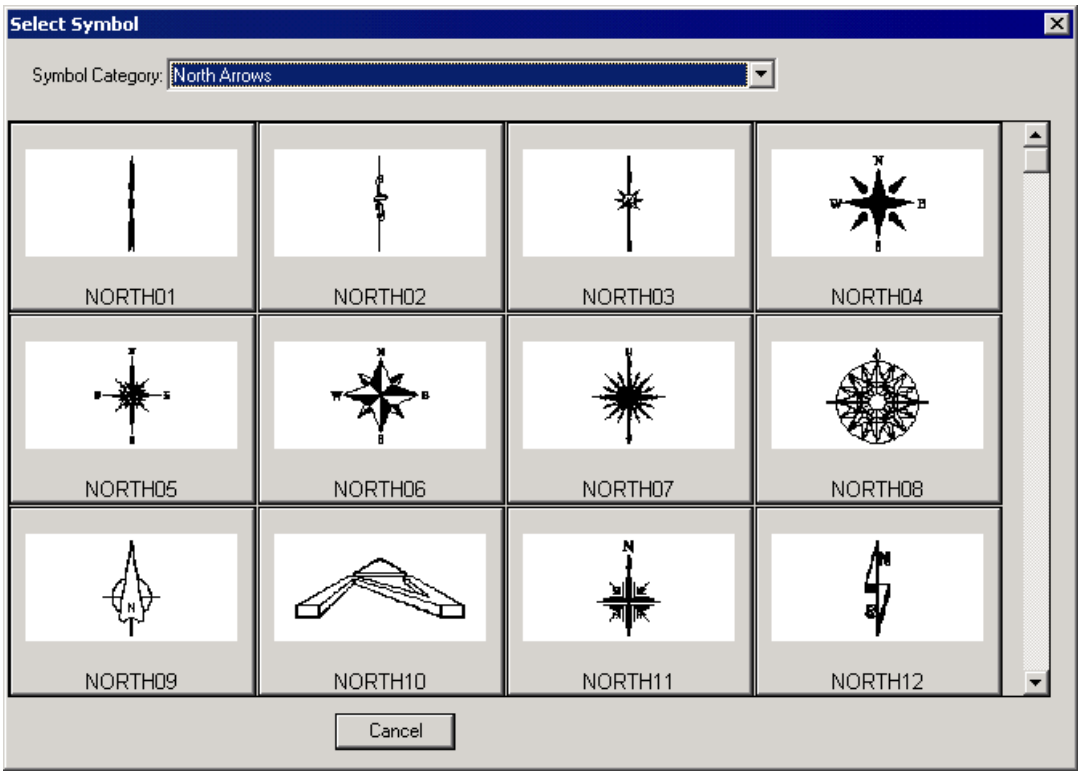
Draw North Arrow

This command inserts a north arrow symbol. You can select from several styles of arrows, and you can add your own by using the Edit Library button which is similar to the Symbols Library command. The north arrow symbol library is stored in the narrow.dta file in the USER folder.



Prompts

Draw North Arrow Dialog *choose an arrow symbol, layer and other variables*
Specify insertion point: *pick a point*
X scale factor <1> / Corner / XYZ: *press Enter*
Y scale factor (default=X): *press Enter*
Rotation angle <0d0'0">: *press Enter*



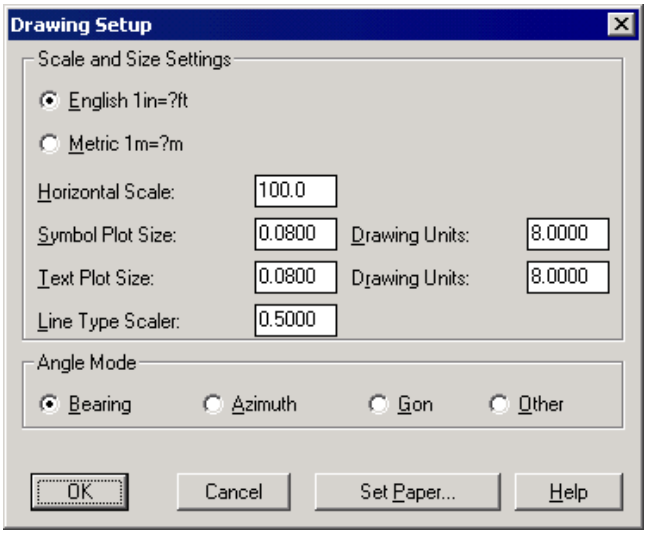
Pulldown Menu Location: Draw
Keyboard Command: narrow
Prerequisite: None

Settings Commands 6

Drawing Setup

Function

This command displays a dialog box for setting drawing parameters, including the plotting scale, size of symbols, label annotation size, and the drawing mode.



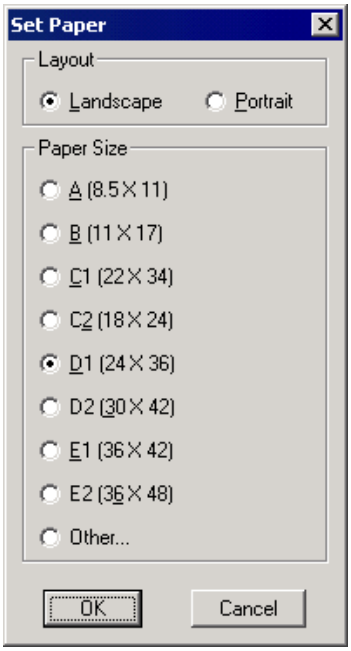
1 Under Scale and Size Settings you can determine scale and size of drawing entities.

- **English 1in=?ft**: This option tells Carlson Survey which unit mode to use. This affects the prompting and reports. When you are working on a drawing in English units, one unit equals one foot. In metric, one unit equals one meter.
- **Metric 1m=?m**: This option sets the metric scale to meters only.
- **Horizontal Scale**: This option allows you to set the horizontal scale of the drawing. For example, if the horizontal scale is set to 50, then 1" = 50' in your drawing.
- **Symbol Plot Size**: This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Symbol Plot Size is not entered in Drawing Units.
- **Drawing Units**: This field shows the result of the Symbol Plot Size value (the scaler) multiplied by the horizontal scale.
- **Text Plot Size**: This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Text Plot Size is not entered in Drawing Units.
- **Drawing Units**: This field shows the result of the Text Plot Size value (the scaler) multiplied by the horizontal scale.
- **Line Type Scaler**: This option sets the linetype scale by multiplying this scaler by the horizontal scale.

2 Under Angle Mode you determine how angles are entered and displayed.

- **Bearing**: This option sets reporting to bearing mode for any of the inquiry commands.
- **Azimuth**: This option sets reporting to north based azimuth mode for any of the inquiry commands.
- **Gon**: This option sets reporting to gon mode for any of the inquiry commands.

3 The Set Paper button allows you to draw a rectangle on the screen that represents the edge of your paper. After you have set the horizontal scale, press the Set Paper button and the following dialog box appears



.

- **Layout**: This option lets you specify landscape or portrait paper orientation. Landscape layout is where the width of the page is greater than the height of the page. Portrait layout is the opposite.

- **Paper Size**: This option allows you to specify the paper size. The numbers in parenthesis represent drawing units and will be multiplied by the horizontal scale to determine the rectangle to be drawn. If you select the Other option, you will be prompted on the command line for the horizontal and vertical sizes of the paper.

Prompts

1 Pick or Type lower left corner point for border <(5000.00 5000.00 0.0)>: **pick point**

2 Erase existing Set Paper boundary [<Yes>/No]? **Y**

This prompt only appears if there is an existing paper boundary in this drawing

3 Set Limits [Yes/<No>]? **Y**

If you answer Yes to Set Limits, drawing limits are enabled, and AgStar restricts the coordinates you can enter to within the paper boundary. Drawing limits also determines the area of the drawing that can display grid dots, and the minimum area displayed by the Zoom All command on the View menu. To turn drawing limits off, type in LIMITS on the command line and set to Off.

Menu Location: Settings

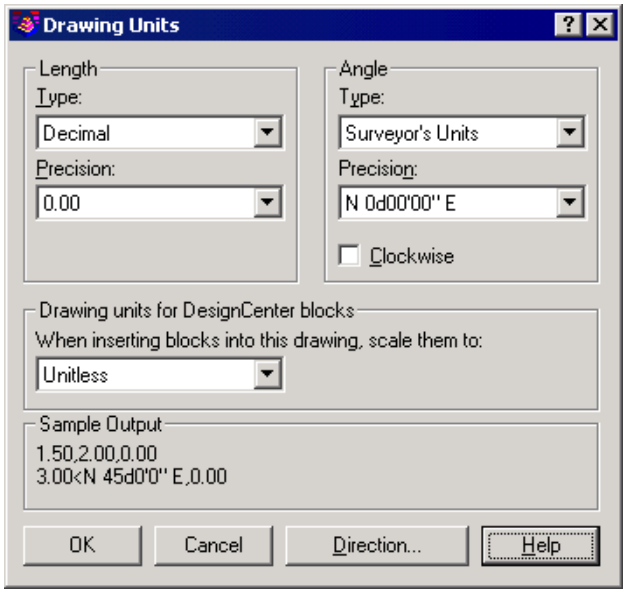
Prerequisite: None

Keyboard Command: SETUP

Units Control

Function

The Drawing Units dialog box controls coordinate and angle display formats and determines precision.



1 Under Length, you specify the current unit of measurement and the precision for the current units.

- **Type**: This field sets the current format for units of measure. The values include Architectural, Decimal, Engineering, Fractional, and Scientific. The Engineering and Architectural formats produce feet-and-inches displays and assume that each drawing unit represents one inch. The other formats can represent any real-world unit.
- **Precision**: This field sets the number of decimal places for the current units display.

2 Under Angle you specify the current angle format and the precision for the current angle display.

- **Type**: This field sets the current angle format.
- **Precision**: This field sets the precision for the current angle display.

Carlson Survey uses the following conventions for the various angle measures: decimal degrees appear as decimal numbers, grads appear with a lowercase g suffix, and radians appear with a lowercase r suffix. The degrees/minutes/seconds format uses d for degrees, ' for minutes, and " for seconds, for example:

123d45'56.7"

Surveyor's units show angles as bearings, using N or S for north or south, degrees/minutes/seconds for how far east or west the angle is from direct north or south, and E or W for east or west, for example:

N 45d0'0" E

The angle is always less than 90 degrees and is displayed in the degrees/minutes/seconds format. If the angle is precisely north, south, east, or west, only the single letter representing the compass point is displayed.

- **Clockwise**: This option calculates positive angles in the clockwise direction. The default direction for positive angles is counterclockwise.

When the program prompts for an angle, you can point in the desired direction or enter an angle regardless of the setting specified for Clockwise.

3 Under Drawing Units for AgStar DesignCenter blocks, you can control the unit of measurement used for block insertions. A block created in units that differ from the units specified in this option is scaled and inserted in the specified units. Select Unitless to insert the block as is and not scale the block to match the specified units. Source content units and Target drawing units settings in the User Preferences tab of the Options dialog box under the Settings menu are used when Insert Units are not defined.

4 Sample Output displays an example of the current settings for units and angles.

Direction displays the Direction Control dialog box described below.

A The Base Angle determines where 0 degrees is located when the program calculates angles. The base angle sets the direction of the base angle. These options affect the entry of angles, object rotation angles, the display format, and the entry of polar, cylindrical, and spherical coordinates. Choose East, North, West, or South, or choose Other to indicate an alternative direction. The default direction for the zero angle is East. In AgStar, the base angle is relative to the orientation of the user coordinate system.

- **East**: Sets the base angle to east (default is zero degrees).
- **North**: Sets the base angle to 90 degrees north.
- **West**: Sets the base angle to 180 degrees west.
- **South**: Sets the base angle to 270 degrees south.
- **Other**: Sets a direction different from the points of the compass.
- **Angle**: Sets the angle. Available only when Other is selected.
- **Pick an Angle**: Uses the pointing device to define the angle based on the angle of an imaginary line connecting any two points you specify. Available only when Other is selected.

Menu Location: Settings

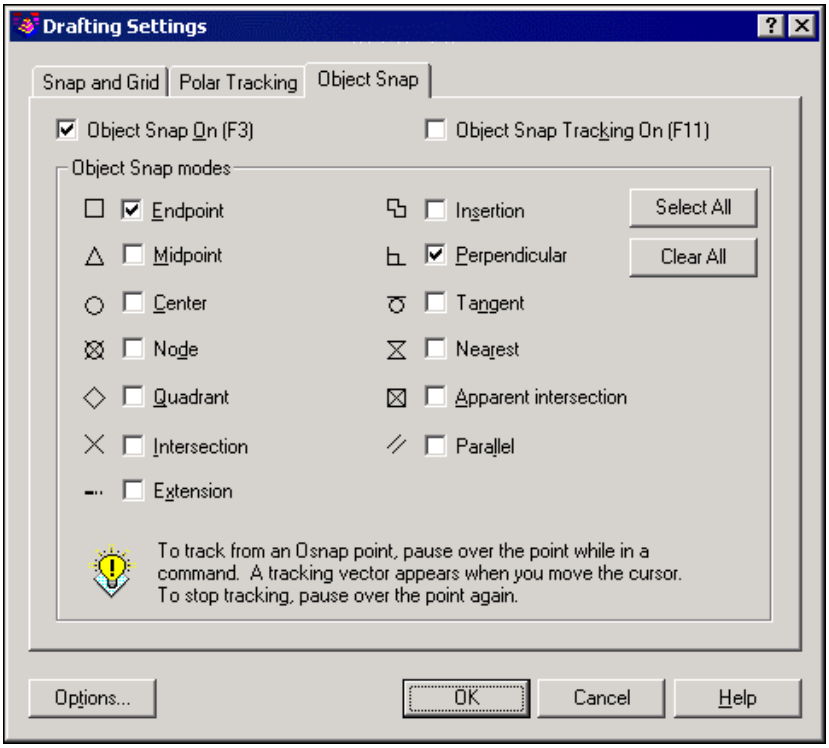
Prerequisite: None

Keyboard Command: UNITS

Object Snap

Function

The Drafting Settings dialog box sets object snap modes.



1 Under Object Snap, you set object snaps.

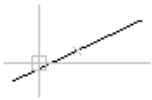
- **Object Snap On**: This option turns running object snaps on and off. The object snaps selected under Object Snap Modes are active while object snap is on. This setting is also controlled by the OSMODE system variable.

- **Object Snap Tracking On**: This option turns object snap tracking on and off. With object snap tracking the cursor can track along alignment paths based on other object snap points when specifying points in a command. To use object snap tracking, you must turn on one or more object snaps.

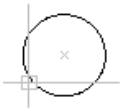
2 Under Object Snap Modes, you turn on running object snaps.

- **Endpoint**: Snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, spline, region, or ray or to the closest corner of a trace, solid, or 3D face.

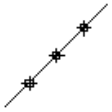
- **Midpoint**: Snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, solid, spline, or xline.



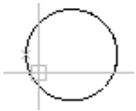
- **Center**: Snaps to the center of an arc, circle, ellipse, or elliptical arc.



- **Node**: Snaps to a point object.

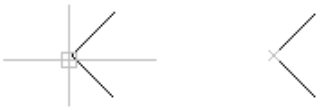


- **Quadrant**: Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.



• **Intersection**: Snaps to the intersection of an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline. Intersection snaps to the edges of regions and curves, but does not snap to the edges or corners of 3D solids.

Extended Intersection snaps to the imaginary intersection of two objects that would intersect if the objects were extended along their natural paths. Carlson Survey automatically turns on Extended Intersection when you select the Intersection object snap mode. You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time. Intersection and Extended Intersection work with edges of regions and curves, but not with edges or corners of 3D solids.



• **Extension**: Causes a temporary extension line to display when you pass the cursor over the endpoint of objects, so you can draw objects to and from points on the extension line.

• **Insertion**: Snaps to the insertion point of an attribute, a block, a shape, or text.

• **Perpendicular**: Snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, solid, spline, or xline. AgStar automatically turns on Deferred Perpendicular snap mode when the object you are drawing requires you to complete more than one perpendicular snap. You can use a line, arc, circle, polyline, ray, xline, multiline, or 3D solid edge as an object from which to draw a perpendicular line. You can use Deferred Perpendicular to draw perpendicular lines between such objects. When the aperture box passes over a Deferred Perpendicular snap point, the program displays a Snaptip and marker.

• **Tangent**: Snaps to the tangent of an arc, circle, ellipse, or elliptical arc. AgStar automatically turns on Deferred Tangent snap mode when the object you are drawing requires you to complete more than one tangent snap. For example, you can use Deferred Tangent to draw a line that is tangent to two arcs, polyline arcs, or circles. When the aperture box passes over a Deferred Tangent snap point, the program displays a marker and Snaptip. If you use the From option in conjunction with the Tangent snap mode to draw objects other than lines from arcs or circles, the first point drawn is tangent to the arc or circle in relation to the last point selected in the drawing area.

• **Nearest**: Snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, spline, or xline.

• **Apparent Intersection**: Apparent Intersection includes two separate snap modes: Apparent Intersection and Extended Apparent Intersection. You can also locate Intersection and Extended Intersection snap points while running Apparent Intersection object snap mode is on. Apparent Intersection snaps to the apparent intersection of two objects (arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline) that do not intersect in 3D space but may appear to intersect in the drawing display. Extended Apparent Intersection snaps to the imaginary intersection of two objects that would appear to intersect if the objects were extended along their natural paths. You might get varying results if you have both the Intersection and Apparent Intersection running object snaps turned on at the same time. Apparent and Extended Apparent Intersection work with edges of regions and curves but not with edges or corners of 3D solids.

• **Parallel**: Draws a vector parallel to another object whenever AgStar prompts you for the second point of a vector. After specifying the first point of a vector, if you move the cursor over a straight line segment of another object,

the program acquires the point. When the path of the object you create is parallel to the line segment, the program displays an alignment path, which you can use to create the parallel object.

- **Clear All**: This option turns off all object snap modes.
- **Select All**: This option turns on all object snap modes.

Menu Location: Settings

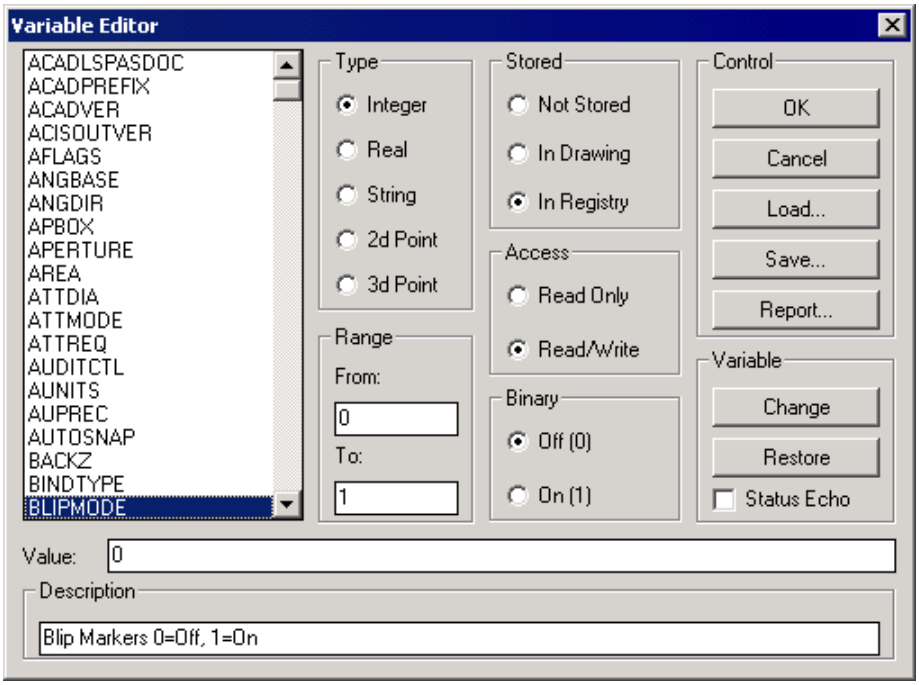
Prerequisite: None

Keyboard Command: OSNAP

Set Environment Variables

Function

The AutoCAD engine stores the values for its operating environment and some of its commands in system variables. Each system variable has an associated type: integer, real, point, switch, or text string. This command allows you to list or change the values of system variables.



1 Dialog Fields

- **List Box**: Contains a list of the variables associated with the currently running version of AutoCAD. There are more items than will display on the list box, use the scroll bar to move up and down through the list. Picking on an item in the list box makes it the current item, causing the information about the item to be displayed, and can be affected by most of the edit commands explained below.
- **Edit Field**: When an item on the list box is picked, its current setting is displayed in the edit field. If you intend to make changes in an item, use standard editing procedures including the use of arrow keys and/or pointer movements to make changes. Once changes have been made, you must use the CHANGE options explained below to effect changes. Pressing enter at the edit field will have no effect on the item in the list. If the item selected is a read-only variable, the edit field will be grayed-out and will not allow input.
- **Description**: When an item on the list box is picked, its definition is referenced and displayed in this field. This can be a benefit in learning the uses of the assorted system variables. This is a display only field, so you can't change the description given.

2 Under Type Group, the type of variable will be displayed indicated by one of the radio buttons. Each of these types are explained below for your benefit. For additional information on variable types used by AutoCAD, obtain and consult a source of AutoCAD documentation.

- **Integer**: Defined as a whole number in the range from -32767 to +32768, no decimal value accepted.
- **Real**: Defined as a real number in the range from -1.797E+308 to +1.797E+308, with extreme decimal accuracy maintained.. Some real variables have a smaller range than previously stated.
- **String**: Defined as a sequential array of characters in the range from 0 to 65535 characters, with a range of ASCII (0-255). Numbers can be included in strings, even though they have no mathematical significance.
- **2D Point**: Defined as a list of two real numbers in the range from -1.797E+308 to +1.797E+308 separated by a comma, having extreme decimal accuracy maintained. Always maintain the X,Y format, one (and only one) comma must be used, separating the X and Y.
- **3D Point**: Defined as a list of three real numbers in the range from -1.797E+308 to +1.797E+308 separated by commas. While editing a 3D point, you must always maintain the X,Y,Z format, two (no less or no more), commas must used, separating the X and Y and Z values.

3 Under Range Group, the variable displayed will usually have a range displayed. The FROM value indicating the minimum, and the TO value being the maximum value accepted.

4 Under the Store Group, depending on the type of variable, AutoCAD may store the value in the drawing or the configuration file, or it may not be stored. Each of these types are explained below for your benefit.

- **Not Stored**: Some variables, such as PLATFORM and CDATE, are not stored because they are system interdependent.
- **In Drawing**: Most variables are stored in the drawing, making the drawing format more personal than just a database of objects. This allows you to open a drawing and have it behave just as though you had never left it.
- **In Config**: These are variables that remain the same regardless of the drawing opened. APERTURE and PICKBOX are just two examples of variables stored in the configuration file.

5 Under Access Group, depending on the type of variable, AutoCAD may not allow you to make changes to it. Each of these types are explained below.

- **Read Only**: Some variables, such as PLATFORM and CDATE, are read-only and therefore cannot be changed. Read-Only variables are marked and the edit field will be grayed indicating that you can't change the variable.
- **Read/Write**: Most variables are read/write and can be changed. These variables are marked and the edit field will be active so you can change the variable.

6 Under Binary Group, depending on the type of variable, the value may be off or on, yes or no. If the variable type is not binary, this group will be grayed out entirely.

- **Off (0)**: Indicate an off condition. Some variables, such as ATTREQ, are simply on or off toggles. You may change a binary item by clicking in this group to change the variable, or changing the value in the edit field.
- **On (1)**: Indicate an on condition. Binary variables are simply on or off toggles. Their range is from 0 to 1. You may change a binary item by clicking to change the variable, or changing the value in the edit field.

7 Control Buttons - These buttons are the main controls in the use of the Variable Editor. Each buttons purpose is explained below.

- **OK**: Used to accept the changes made during the variable editing process, returning you to the command prompt with changes in effect.
- **Cancel**: Used to cancel the changes made during the variable editing process, returning you to the command prompt without the changes in effect.
- **Load**: Used to load a saved set of system variables. This allows you to create a drawing, save the system variables, open a second drawing, and load those variables into that drawing. Read-only variables are skipped.

- **Save**: Used to save the current system variables to a disk file. All system variables are stored to the file, even those that are marked as read-only.
- **Print**: Used to print the current system variables. After choosing this option, you will prompted for an output filename, then the program will proceed to write the system variables to the file. This file can be loaded into any editor or word processor, edited and printed.

8 Variable Buttons - These buttons are used to control the changes in variables, while using the Variable Editor. Each buttons purpose is explained below.

- **Change**: Used to execute the changes typed into the edit field. You must use this button, simply pressing enter will not make the change.
- **Restore**: Used to cancel the changes typed into the edit field. If you make a mistake or change your mind while making changes in the edit field, press this button to restore the edit field to the value before editing.
- **Status**: Used to determine if the program will echo the status of changes being made to the command area. If this toggle is on, any changes made from the dialog will echo the change. Also if a stream of change commands is being read from a file, and the toggle is on, the changes taking place will be displayed.

Note: This command displays many more system variables than are found in Chapter 18, which contains a list of **supported** system variables. Modification of any system variable other than the supported ones found in Chapter 18 is done at your own risk, and may result in program errors requiring a re-installation of AgStar.

Menu Location: Settings

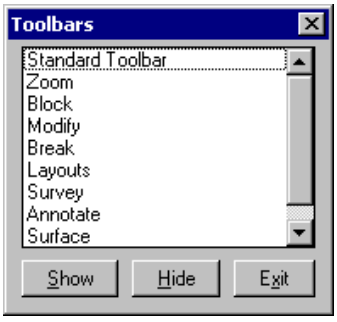
Prerequisite: None

Keyboard Command: VAREDIT

Toolbars

Function

This command allows you to display and hide toolbars. Click on a toolbar name and press the Show or Hide button.



- **Show**: Turns on the selected toolbar. If the toolbar is already visible, then this does nothing.
- **Hide**: Turns off the selected toolbar. If the toolbars is already hidden, then this does nothing. If the toolbar is floating, you can also turn it off by clicking the x in the upper right corner.
- **Exit**: Exits this command

Menu Location: Settings

Prerequisite: None

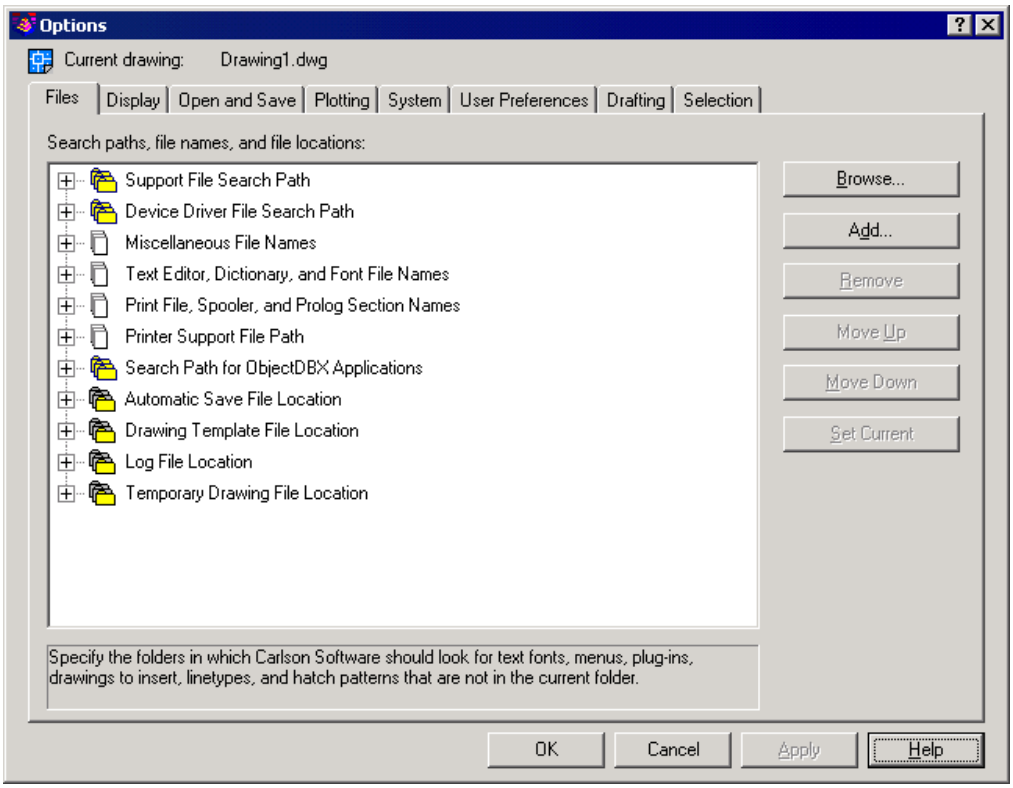
Keyboard Command: TBARCFG

Options

Function

This command allows you to customize the AgStar settings.

Files Tab



Under the Files Tab, you specify the directories in which the program searches for support, driver, menu, and other files. You can also specify optional, settings such as which dictionary to use for checking spelling. Each option under the Files Tab displays a list of the directories and files used by Carlson Survey. To specify a location for a directory or file, double-click the directory or file from the list. Choose Browse and use the Browse for Folder dialog box (a standard file selection dialog box) to locate the directory or file you want to use.

1 **Support File Search Path**: Specifies the directories in which Carlson Survey searches for support files. In addition to the files required to run the program, you can include files for fonts, menus, drawings to insert, linetypes, and hatch patterns in the support file search path.

2 **Device Driver File Search Path**: Specifies where the program looks for device drivers for the video display, pointing devices, printers, and plotters.

3 **Miscellaneous File Names**: Specifies the names and locations of various types of files.

- **Menu File**: Specifies the location of the menu file.
- **Default Internet Location**: Specifies the default Internet location used by both the Connect to Internet option on the Help menu, and the Launch Browser button on the Standard toolbar.

4 **Text Editor, Dictionary, and Font File Names**: Specifies a number of optional settings.

- **Text Editor Application**: Specifies the text editor application to use for editing mtext objects.
- **Custom Dictionary File**: Specifies a custom dictionary to use (if you have one).
- **Alternate Font File**: Specifies the location of the font file to use if Carlson Survey cannot locate the original font and an alternate font is not specified in the font mapping file. If you choose Browse, the program displays the

Alternate Font dialog box, from which you can choose an available font.

- **Font Mapping File**: Specifies the location of the file that defines how Carlson Survey should convert fonts it cannot locate.

5 Print File, Spooler, and Prolog Section Names: Specifies settings related to plotting.

- **Plot File Name For Legacy Plotting Scripts**: Specifies a default name for the temporary plot files used with plotting scripts created with earlier versions of Autodesk products. The default name is the drawing name plus .plt file name. The default name used with Carlson Survey drawings is the drawing name-layout name plus the .plt file name extension. Some plotting device drivers, however, use a different plot file extension name. This option only affects the default plot file name used for plotting scripts created with earlier versions of Autodesk products.

- **Print Spool Executable**: Specifies the application to use for print spooling. You can enter the executable file name as well as any command line arguments you want to use. For example, you can enter myspool.bat %s to spool plot files to myspool.bat and have a unique plot file name automatically generated.

6 **Printer Support File Path**: Specifies search path settings for printer support files.

- **Print Spooler File Location**: Specifies the path for print spool files. AgStar writes the plot to this location.
- **Printer Configuration Search Path**: Specifies the path for printer configuration files (PC3 files).
- **Printer Description File Search Path**: Specifies the path for files with a .pmp file extension, or printer description files.
- **Plot Style Table Search Path**: Specifies the path for files with an .stb or .ctb extension, or plot style table files (both named plot style tables and color-dependent plot style tables).

7 **Search Path for ObjectDBX Applications**: Specifies the path for ObjectDBX™ application files. You can enter multiple URL addresses under this option. (Multiple URLs should be separated by a semi-colon.) AgStar searches the specified URLs when an associated ObjectDBX application cannot be located. Only URL addresses can be entered in this option.

8 **Automatic Save File Location**: Specifies the path for the file created when you select Automatic Save on the Open and Save tab.

9 **Drawing Template File Location**: Specifies the path for the template files used by the setup wizards.

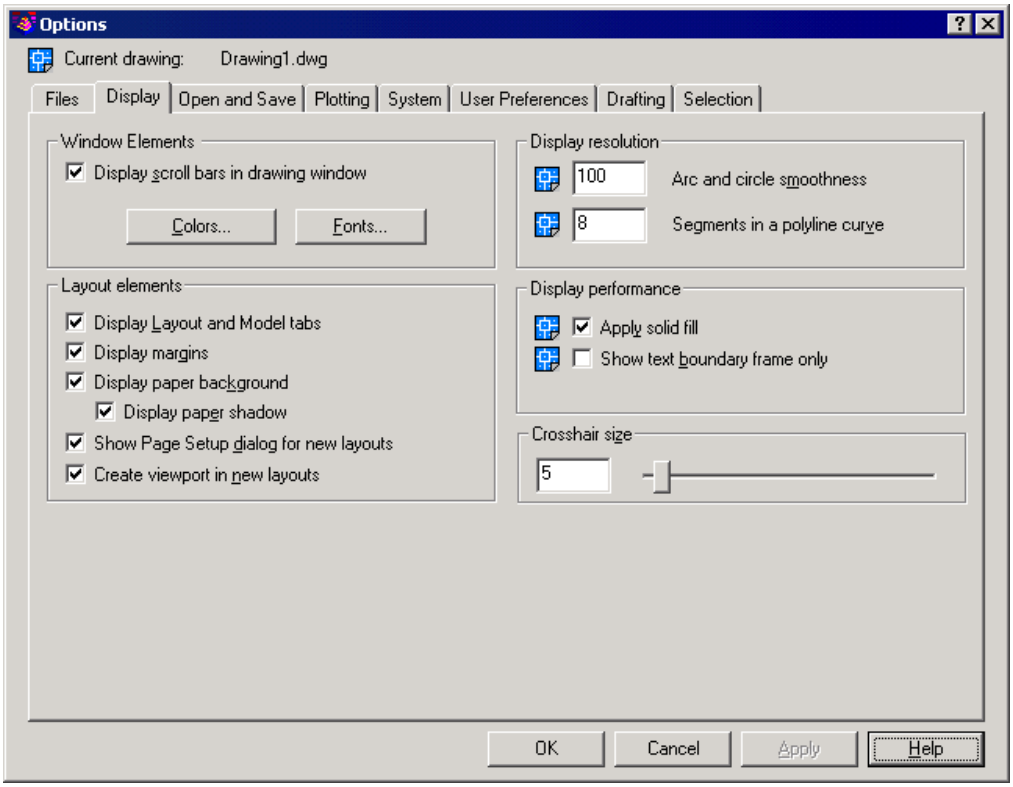
10**Log File Location**: Specifies the path for the log file created when you select Maintain a Log File on the Open and Save tab.

11**Temporary Drawing File Location**: Specifies the location AgStar uses to store temporary files. The program creates temporary files on disk and then deletes them when you exit the program. If you plan to run the program from a write-protected directory (for example, if you are working on a network or opening files from a CD), specify an alternate location for your temporary files. The directory you specify must not be write-protected.

12**Buttons**: You use the following buttons to manipulate the files and paths.

- **Browse**: Displays the Browse for Folder or Select a File dialog box, depending on what you selected in the List of Folders and Files.
- **Add**: Adds a search path for the selected directory.
- **Remove**: Removes the selected search path or file.
- **Move Up**: Moves the selected search path above the preceding search path.
- **Move Down**: Moves the selected search path below the following search path.
- **Set Current**: Makes the selected project or spelling dictionary current.

Display Tab



1 Under Window Elements, you control display settings specific to the Carlson Survey drawing environment.

- **Display Scroll Bars in Drawing Window**: Specifies whether to display scroll bars at the bottom and right sides of the drawing area.
- **Colors**: Displays the Color Options dialog box. Use this dialog box to specify the colors of elements in the window.
- **Fonts**: Displays the Command Line Window Font dialog box. Use this dialog box to specify the font for the command line text.

2 Under Layout Elements, you control options for existing and new layouts. A layout is a paper space environment in which you can set up drawings for plotting.

- **Display Layout and Model Tabs**: Specifies whether to display the layout and Model tabs at the bottom of the drawing area.
- **Display Margins**: Specifies whether margins are displayed in a layout. Margins appear as dashed lines. Objects drawn outside of the margins are clipped or omitted when the drawing is plotted.
- **Display Paper Background**: Specifies whether a representation of the specified paper size is displayed in a layout. The paper size and plot scale determine the size of the paper background.
- **Display Paper Shadow**: Specifies whether a shadow is displayed around the paper background in a layout.
- **Show Page Setup Dialog for New Layouts**: Specifies whether the Page Setup dialog box is displayed when you create a new layout. Use this dialog box to set options related to paper and plot settings.
- **Create Viewport in New Layouts**: Specifies whether a viewport is created when you create a new layout.

3 Under Display Resolution, you control the quality of the display of objects. If you set high values to improve display quality, the impact on performance is significant.

- **Arc and Circle Smoothness**: Controls the smoothness of circles, arcs, and ellipses. A higher number produces smoother objects, but requires more time to regenerate, pan, and zoom the objects. You can improve performance by setting this option to a low value such as 100 for drawing, and increasing the value for rendering. The valid range is 1 to 20,000. The default setting is 100. This setting is saved in the drawing. To change the default for new drawings, consider specifying this setting in the template files on which you base your new drawings.

• **Segments in a Polyline Curve**: Sets the number of line segments to be generated for each polyline curve. The higher the number, the greater the performance impact. Set this option to a low value such as 4 to optimize performance for drawing. Values range from -32767 to 32767. The default setting is 8. This setting is saved in the drawing.

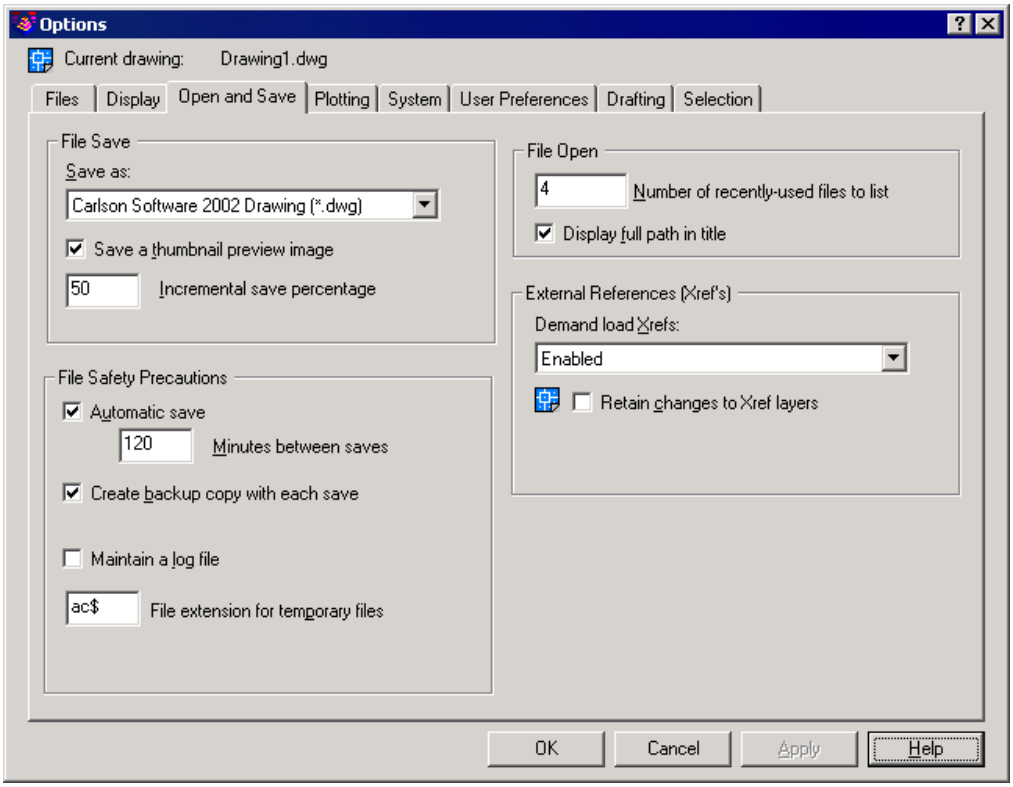
4 Under Display Performance, you control display settings that affect Carlson Survey performance.

• **Apply Solid Fill**: Controls whether solid fills in objects are displayed. Objects with solid fill include multilines, traces, solids, all hatches (including solid-fill), and wide polylines. You must regenerate the drawing by using REGEN for this setting to take effect. This setting is saved in the drawing. Clear this option to optimize performance.

• **Show Text Boundary Frame Only**: Displays the frames for text objects instead of displaying the text objects. After you select or clear this option, you must use REGEN to update the display. This setting is saved in the drawing. Select this option to optimize performance.

5 Under Crosshair Size, you control the size of the crosshairs. The valid range is from 1 to 100 percent of the total screen. At 100 percent, the ends of the crosshairs are never visible. When the size is decreased to 99 percent or below, the crosshairs have a finite size, and the ends of the crosshairs are visible at the edge of the drawing area. The default size is 5 percent.

Open and Save Tab



Under the Open and Save Tab, you control options that relate to opening and saving files.

1 Under File Save, you control settings related to saving a file in Carlson Survey.

• **Save As**: Displays the valid file formats used when saving a file with SAVE and SAVEAS. The file format selected for this option is the default format that all drawings are saved as when you use SAVE or SAVEAS. Saving an AgStar file to any DXF format affects performance. Set the Save As option to AgStar 2000 Drawing to optimize performance while saving.

• **Save a Thumbnail Preview Image**: Specifies whether an image of the drawing should be displayed in the Preview area of the Select File dialog box.

• **Incremental Save Percentage**: Sets the percentage of potential wasted space in a drawing file. When the

specified percentage is reached, the program performs a full save instead of an incremental save. Full saves eliminate wasted space. If you set Incremental Save Percentage to 0, every save is a full save. Although incremental saves increase the size of your drawing, avoid setting a very low value. Low values degrade performance because the program performs time-consuming full saves more often. For optimum performance, set the value to 50. If hard disk space is scarce, set the value to 25. If you set the value to 20 or less, performance of the SAVE and SAVEAS commands slows significantly.

2 File Safety Precautions settings help you avoid data loss and detect errors.

- **Automatic Save**: Saves your drawing automatically at the interval you specify. You can specify the location of all Autosave files by using the SAVEFILEPATH system variable. SAVEFILE (read-only) stores the name of the Autosave file.

- **Minutes Between Saves**: Specifies how often the drawing is saved when using Automatic Save. The value is stored in SAVETIME.

- **Create Backup Copy with Each Save**: Specifies whether a backup copy of a drawing is created when you save the drawing. The backup copy is created in the same location as the drawing. The ISAVEBAK system variable controls whether a backup copy of the drawing is created.

- **Maintain a Log File**: Specifies whether the contents of the text window are written to a log file. To specify the location and name of the log file, use the Files tab in the Options dialog box.

- **File Extension for Temporary Files**: Specifies a unique extension that allows you to identify your temporary files in a network environment. The default extension is .ac$.

3 Under File Open, you control settings that relate to recently used files and open files.

- **Number of Recently Used Files to List**: Controls the number of recently used files that are listed in the File menu for quick access. Valid values are 0 to 9.

- **Display Full Path In Title**: Displays the full path of the active drawing in the drawing's title bar, or in the Carlson Survey title bar if the drawing is maximized.

4 Under External References (Xrefs), you control the settings that relate to editing and loading external references.

- **Demand Load Xrefs**: Controls demand loading of xrefs. Demand loading improves performance by loading only the parts of the referenced drawing needed to regenerate the current drawing. External Reference File Demand Load is also controlled by the XLOADCTL system variable.

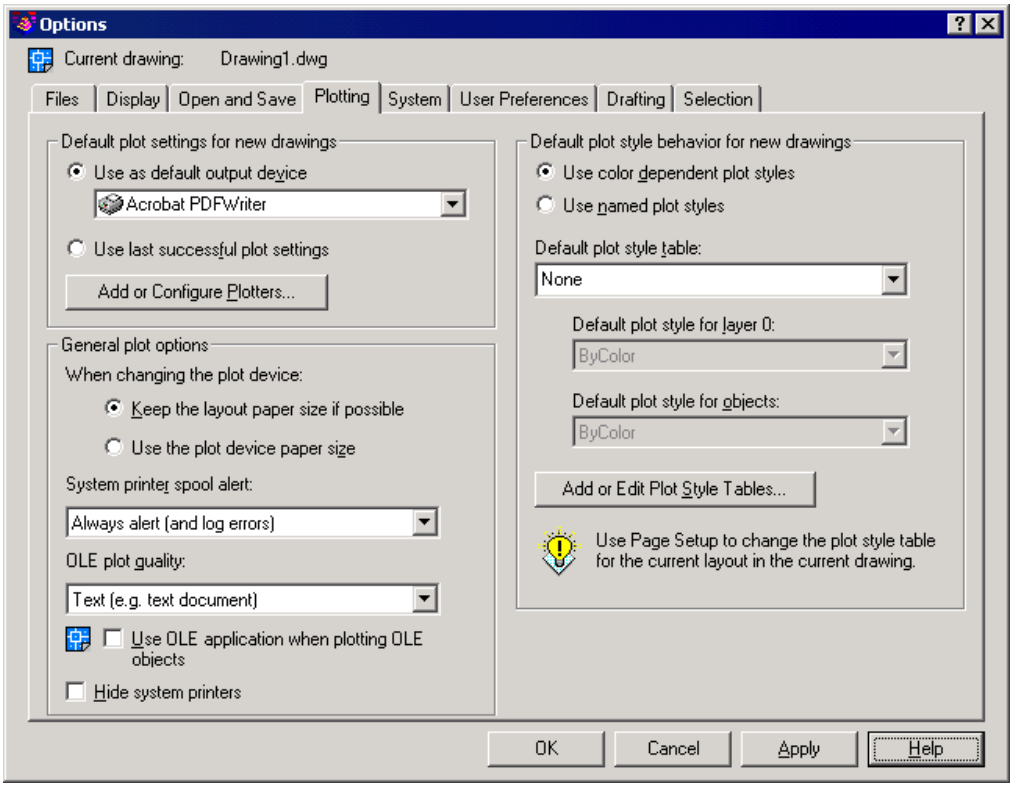
- **Disabled**: Turns off demand loading.

- **Enabled**: Turns on demand loading and improves performance. Select the Enabled setting to enhance the loading process when you are working with clipped xrefs that contain a spatial or layer index. When this option is selected, other users cannot edit the file while it is being referenced.

- **Enabled with Copy**: Turns on demand loading but uses a copy of the referenced drawing. Other users can edit the original drawing.

- **Retain Changes to Xref Layers**: Saves changes to layer properties and states for xref-dependent layers. When the drawing is reloaded, the properties currently assigned to xref-dependent layers are retained. This setting is saved in the drawing.

Plotting Tab



Under the Plotting Tab, you control options related to plotting.

1 Under Default Plot Settings For New Drawings, you control default plotting settings for new drawings or drawings created in AutoCAD Release 14 or earlier that have never been saved in Carlson Survey 2000 format.

- **Use As Default Output Device**: Sets the default output device for new drawings and for drawings created in AutoCAD Release 14 or earlier that have never been saved in Carlson Survey 2000 format. The list displays any plotter configuration files (PC3) found in the plotter configuration search path and any system printers that are configured in the system.
- **Use Last Successful Plot Settings**: Sets the plotting settings according to the settings of the last successful plot.
- **Add or Configure Plotters**: Displays the Autodesk Plotter Manager (a Windows system window). You can add or configure a plotter with the Autodesk Plotter Manager.

2 Under General Plot Options, you control options that relate to the general plotting environment including paper size settings, system printer alert behavior, and OLE objects in an AgStar drawing.

- **Keep the Layout Paper Size If Possible**: Uses the paper size specified on the Layout Settings tab in the Page Setup dialog box under the File menu as long as the selected output device can plot to this paper size. If the selected output device cannot plot to this paper size, the program displays a warning message and uses the paper size specified either in the plotter configuration file (PC3) or in the default system settings if the output device is a system printer. You can also set Keep the Layout Paper Size If Possible by setting PAPERUPDATE to 0.
- **Use the Plot Device Paper Size**: Uses the paper size specified either in the plotter configuration file (PC3) or in the default system settings if the output device is a system printer. You can also set Use the Plot Device Paper Size by setting PAPERUPDATE to 1.
- **System Printer Spool Alert**: Determines whether to alert you if the plotted drawing is spooled through a system printer because of an input or output port conflict.
- **Always Alert (And Log Errors)**: Alerts you and always logs an error when the plotted drawing spools through a system printer.
- **Alert First Time Only (And Log Errors)**: Alerts you once and always logs an error when the plotted drawing spools through a system printer.

- **Never Alert (And Log First Error)**: Never alerts you and logs only the first error when the plotted drawing spools through a system printer.

- **Never Alert (Do Not Log Errors)**: Never alerts you or logs an error when the plotted drawing spools through a system printer.

- **OLE Plot Quality**: Determines the quality of plotted OLE objects. The values are Line Art, Text, Graphics, Photograph, and High Quality Photograph.

- **Use OLE Application When Plotting OLE Objects**: Launches the application used to create the OLE object when plotting a drawing with OLE objects. You can use this option if you want to optimize the quality of plotted OLE objects. This setting is saved in the drawing. You can also control this option by using the OLESTARTUP system variable.

- **Hide System Printer**: Controls whether Windows system printers are displayed in the Plot and Page Setup dialog boxes under the File menu. This option hides standard Windows system printers only. You can control the size of the list of devices in the Plot and Page Setup dialog boxes by moving a device's PC3 file out of the Plotters directory and its subdirectories.

3 Under Default Plot Style Behavior, you control options related to plot style behavior in all drawings. Changing the default plot style behavior using the Options dialog box does not affect the current drawing. A plot style is a collection of property settings defined in a plot style table and applied when the drawing is plotted. The default setting is Use Color Dependent Plot Styles. The plot style list on the Object Properties toolbar is disabled by default. You enable the list after you select the Use Named Plot Styles option and open a new drawing. You can also control Default Plot Style Behavior by using the PSTYLEPOLICY system variable.

- **Use Color Dependent Plot Styles**: Uses color-dependent plot styles in both new drawings and drawings created in earlier versions of Autodesk products. Color-dependent plot styles use the numbers from the color index to create a plot style table with a .ctb file extension. Each color is defined by a name or number ranging from 1 to 255. You can assign each color number to a different pen on a pen plotter to achieve different property settings in the plotted drawing. If this option is selected, a plot style is created for each color setting. You can also control Use Color Dependent Plot Styles by setting the PSTYLEPOLICY system variable to 1.

If you want to change the default plot style behavior for a drawing, select this option or Use Named Plot Styles before opening or creating a drawing. Changing the default plot style behavior using the Options dialog box affects only new drawings or drawings created in an earlier release of an Autodesk product that have never been saved in AgStar 2000 format. This setting is saved with the drawing. Once a drawing is saved with Use Color Dependent Plot Styles as the default, you can change the default to Use Named Plot Styles with a migration utility with a migration utility. However, once a drawing is saved with Use Named Plot Styles as the default, you cannot change it to Use Color Dependent Plot Styles.

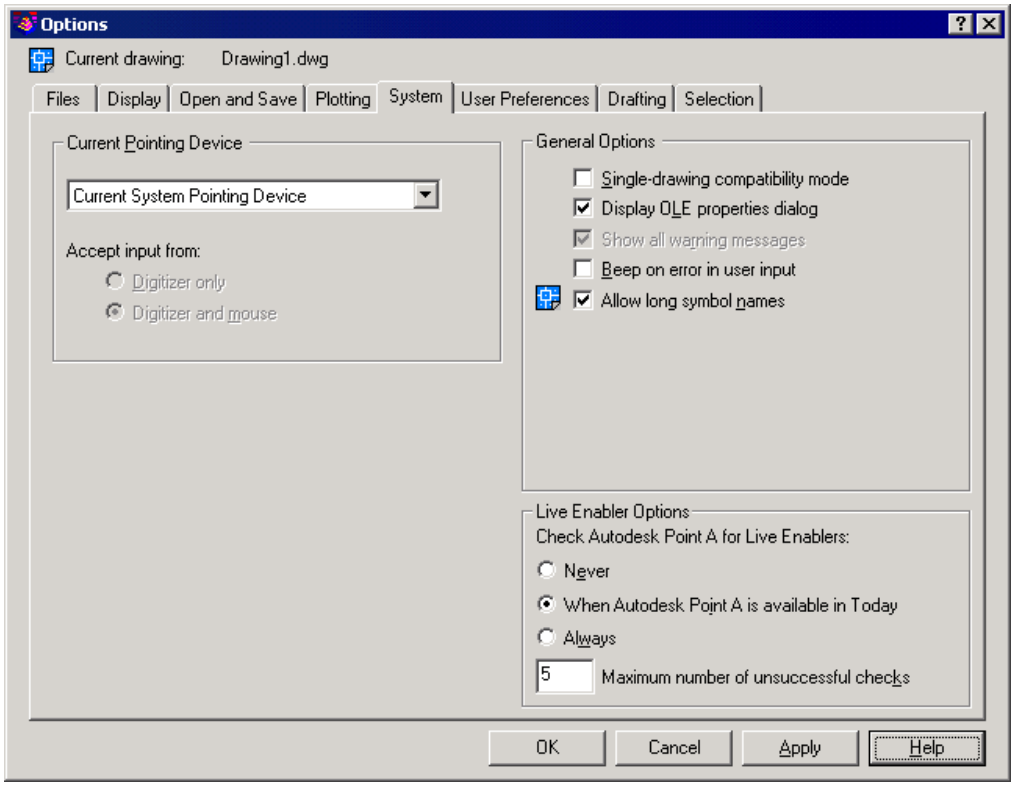
- **Use Named Plot Styles**: Uses named plot styles in both new drawings and drawings created in earlier versions of Autodesk products. AgStar plots the drawing according to the property settings you specify in the plot style definition. The plot style is defined in the plot style table attached to the layout or viewport. Named plot style tables are files with the file extension .stb. You can also control Use Named Plot Styles by setting the PSTYLEPOLICY system variable to 0.

If you want to change the default plot style behavior for a drawing, select this option or Use Color Dependent Plot Styles before opening or creating a drawing. Changing the default plot style behavior using the Options dialog box affects only new drawings or drawings created in an earlier release of an Autodesk product that have never been saved in AgStar 2000 format. This setting is saved with the drawing. Once a drawing is saved with Use Color Dependent Plot Styles as the default, you can change the default to Use Named Plot Styles with a migration utility. However, once a drawing is saved with Use Named Plot Styles as the default, you cannot change it to Use Color Dependent Plot Styles.

- **Default Plot Style Table**: Specifies the default plot style table to attach to new drawings. A plot style table is a file with a .ctb or an .stb extension that includes and defines plot styles. If you are using color-dependent plot styles, this option lists all color dependent plot style tables found in the search path as well as the value of None. If you are using named plot styles, this option lists all named plot styles tables.

- **Default Plot Style for Layer 0**: Sets the default plot style for Layer 0 for new drawings or drawings created with earlier releases of an Autodesk product that have never been saved in Carlson Survey 2000 format. The list displays the default value Normal and alphabetically displays any plot styles defined in the currently loaded plot style table.
- **Default Plot Style for Objects**: Sets the default plot style that is assigned when you create new objects. The list displays a BYLAYER, BYBLOCK, and Normal style, and it alphabetically displays any plot styles defined in the currently loaded plot style table.
- **Add or Edit Plot Style Tables**: Displays the Autodesk Plot Style Table Manager (a Windows Explorer window). You can create or edit plot style tables with the Autodesk Plot Style Table Manager.

System Tab



Under the System Tab, you control AgStar system settings.

1 Under Current Pointing Device, you control options that relate to the pointing device. This field displays a list of the available pointing device drivers.

- **Current System Pointing Device**: Sets the system pointing device as current.
- **Wintab Compatible Digitizer**: Sets the Wintab Compatible Digitizer as current.
- **Accept Input From**: Specifies whether the program accepts input from both a mouse and a digitizer or ignores mouse input when a digitizer is set.

2 Under General Options, you control general options that relate to system settings.

- **Single-Drawing Compatibility Mode**: Specifies whether a single-drawing interface (SDI) or a multi-drawing interface (MDI) is enabled. If you select this option, Carlson Survey opens only one drawing at a time. If you clear this option, the program can open multiple drawings at once.
- **Display OLE Properties Dialog**: Controls the display of the OLE Properties dialog box when inserting OLE objects into a drawing.
- **Show All Warning Messages**: Displays all dialog boxes that include a Don't Display This Warning Again option. All dialog boxes with warning options are displayed regardless of previous settings specific to each dialog box.

- **Beep on Error in User Input**: Specifies whether the program should sound an alarm beep when it detects an invalid entry.

- **Allow Long Symbol Names**: Determines whether long symbol names are enabled. Named objects can include up to 255 characters. Names can include letters, numbers, blank spaces, and any special character not used by Windows and Carlson Survey for other purposes. When this option is enabled, long names can be used for layers, dimension styles, blocks, linetypes, text styles, layouts, UCS names, views, and viewport configurations. This option is saved in the drawing.

3 Under Live Enabler Options, you specify how Carlson Survey checks for Object Enablers. Using Object Enablers, you can display and use custom objects in Carlson Survey drawings even when the ObjectARX application that created them is unavailable.

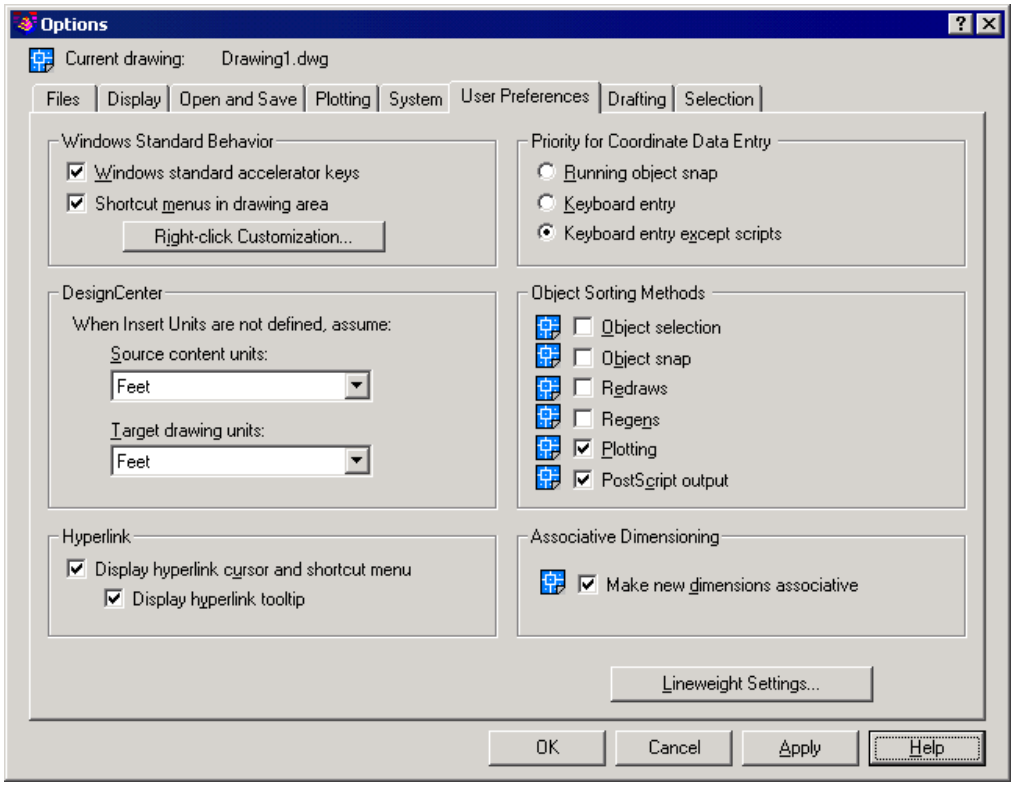
- **Never**: Prevents the program from checking for Object Enablers regardless of your settings in the Today window.

- **When Autodesk Point A is Available in Today**: Carlson Survey checks for Object Enablers only if Autodesk Point A is open in the Today window. It is not necessary for the Today window to be open. However, the program checks for Object Enablers only if a live Internet connection is present.

- **Always**: Carlson Survey always checks for Object Enablers regardless of your settings in the Today window.

- **Maximum Number of Unsuccessful Checks**: Specifies the number of times AgStar will continue to check for Object Enablers after unsuccessful attempts.

User Preferences Tab



Under the User Preferences Tab, you control options that optimize the way you work in Carlson Survey.

1 Under Windows Standard Behavior you specify whether Windows behavior is applied when working in AgStar.

- **Windows Standard Accelerator Keys**: Follows Windows standards in interpreting keyboard accelerators (for example, CTRL+C equals COPYCLIP). If this option is cleared, the program interprets keyboard accelerators by using AgStar standards rather than Windows standards (for example, CTRL+C equals Cancel, CTRL+V toggles among the viewports).

• **Shortcut Menus in Drawing Area**: Controls whether right-clicking in the drawing area displays a shortcut menu or issues ENTER.

• **Right-Click Customization**: Displays the Right-Click Customization dialog box. You can also set Shortcut Menus in Drawing Area and the right-click customization settings by using the SHORTCUTMENU system variable.

2 DesignCenter - this section does not apply to Carlson Survey.

3 Under Hyperlink, you control settings that relate to the display properties of hyperlinks.

• **Display Hyperlink Cursor and Shortcut Menu**: Controls the display of the hyperlink cursor and shortcut menu. The hyperlink cursor appears alongside the crosshairs whenever the pointing device moves over an object that contains a hyperlink. The hyperlink shortcut menu provides additional options when you select an object that contains a hyperlink and then right-click in the drawing area. If this option is cleared, the hyperlink cursor is never displayed and the Hyperlink option on shortcut menus is not available (if shortcut menus are enabled).

• **Display Hyperlink Tooltip**: Controls the display of the Hyperlink tooltip. If this option is selected, a hyperlink tooltip is displayed when the pointing device moves over an object that contains a hyperlink. Display Hyperlink Cursor and Shortcut Menu must be selected to enable this option

4 Under Priority for Coordinate Data Entry, you control how Carlson Survey responds to input of coordinate data.

• **Running Object Snap**: Specifies that running object snaps override coordinate entry at all times. You can also set OSNAPCOORD to 0 to enable Running Object Snap.

• **Keyboard Entry**: Specifies that coordinate entry overrides running object snaps at all times. You can also set OSNAPCOORD to 1 to enable Keyboard Entry.

• **Keyboard Entry Except Scripts**: Specifies that coordinate entry overrides running object snaps, except in scripts. You can also set OSNAPCOORD to 2 to enable Keyboard Entry Except Scripts.

5 Under Object Sorting Methods, you determine the sort order of objects.

• **Object Selection**: Controls how objects are sorted during selection. If this option is selected, Carlson Survey sorts objects available for selection from those created first to those created last. If two overlapping objects are chosen during object selection, the program recognizes the newest object as the selected object. If this option is cleared, object selection is determined by a random sort order. This setting is saved in the drawing.

• **Object Snap**: Controls how objects are sorted when using Object Snap. If this option is selected, Carlson Survey sorts objects available for selection from those created first to those created last. If two overlapping objects are chosen when using Object Snap, the program recognizes the newest object as the object to snap to. If this option is cleared, Object Snap is determined by a random sort order. This setting is saved in the drawing.

• **Redraws**: Controls how objects are sorted when using the REDRAW command. If this option is selected, the program sorts and redraws objects in the drawing from those created first to those created last. If this option is cleared, the redrawing of objects is determined by a random sort order. This setting is saved in the drawing.

• **Regens**: Controls how objects are sorted when using the REGEN command. If this option is selected, the program sorts and regenerates objects in the drawing from those created first to those created last. If this option is cleared, the regeneration of objects is determined by a random sort order. This setting is saved in the drawing.

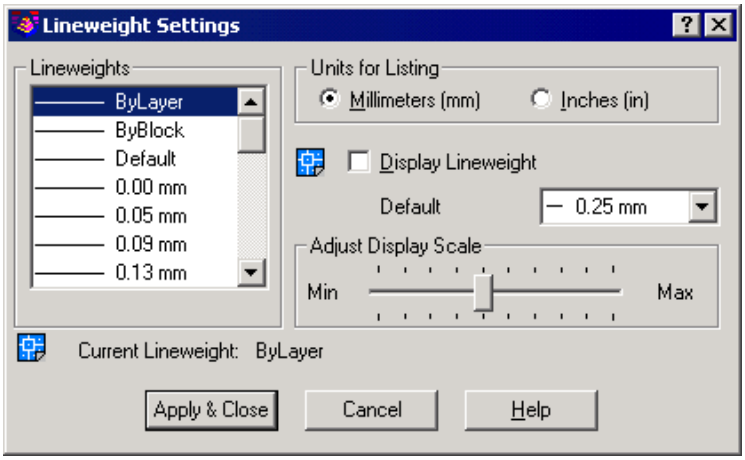
• **Plotting**: Controls how objects are sorted during plotting. If this option is selected, the program sorts and plots objects in the drawing from those created first to those created last. If this option is cleared, the plotting of objects is determined by a random sort order. This setting is saved in the drawing.

• **PostScript Output**: Controls how objects are sorted in PostScript output. If this option is selected, the program sorts and exports objects in the drawing from those created first to those created last. If this option is cleared, the exporting of objects is determined by a random sort order. This setting is saved in the drawing.

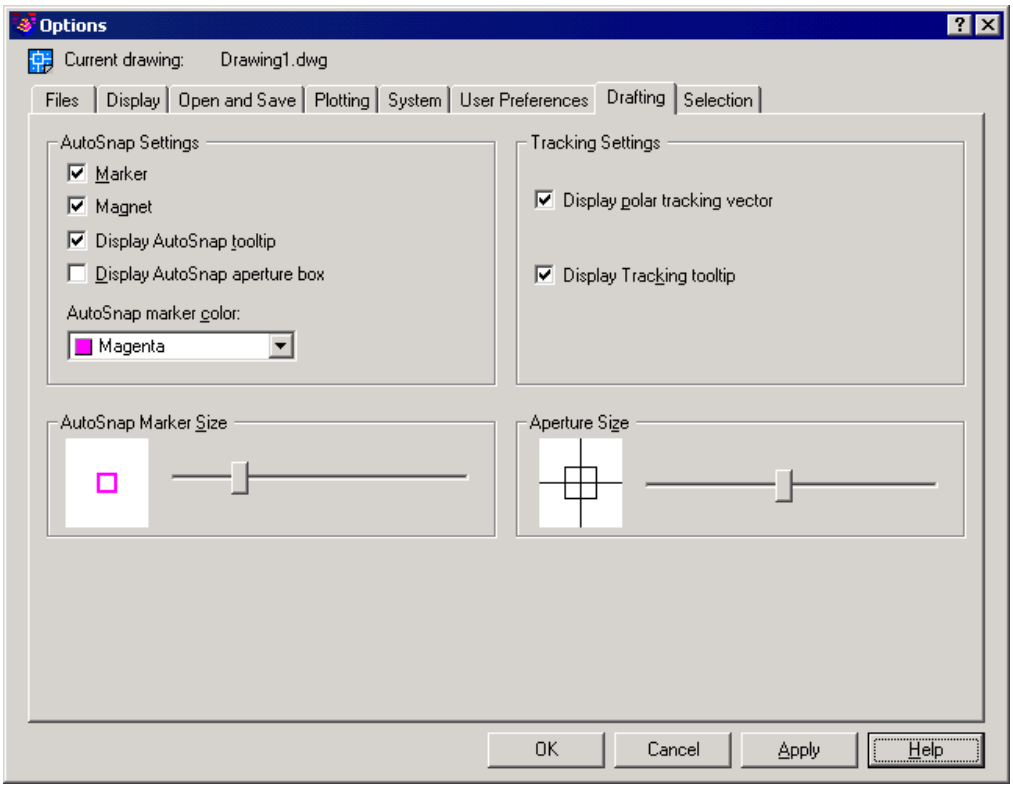
• **Lineweight Settings**: Displays the Lineweight Settings dialog box. Use this dialog box to set lineweight options, such as display properties and defaults, and also to set the current lineweight.

6 Associative Dimensioning - this section does not apply to Carlson Survey.

7 Lineweight Settings - Displays the Lineweight Settings dialog box. Use this dialog box to set lineweight options, such as display properties and defaults, and also to set the current lineweight.



Drafting Tab



Under the Drafting Tab, you specify a number of general editing options.

1 Under AutoSnap Settings, you control settings that relate to object snaps. Using object snaps, you can locate exact points and planes including endpoints, midpoints, centers, nodes, quadrants, intersections, insertion points, and perpendicular and tangent planes.

- **Marker**: Controls the display of the AutoSnap™ marker. The marker is a geometric symbol that displays the object snap location when the crosshairs move over a snap point on an object. You can also enable the Marker by setting AUTOSNAP to 1.
- **Magnet**: Sets the AutoSnap magnet on or off. The magnet is an automatic movement of the crosshairs that locks the crosshairs onto the nearest snap point. You can also enable the Magnet by setting AUTOSNAP to 4.
- **Display AutoSnap Tooltip**: Controls the display of the AutoSnap tooltip. The tooltip is a text flag that describes

which part of the object you are snapping to. You can turn object snaps on and off from the Object Snap tab in the Drafting Settings dialog box. You can also enable the Display AutoSnap tooltip option by setting AUTOSNAP to 2.

- **Display AutoSnap Aperture Box**: Controls the display of the AutoSnap aperture box. The aperture box is a box that appears inside the crosshairs when you snap to an object. You can also set the Display AutoSnap Aperture Box by using the APBOX system variable.

- **AutoSnap Marker Color**: Specifies the color of the AutoSnap marker.

2 Under AutoSnap Marker Size, you set the display size for the AutoSnap marker. Values range from 1 to 20 pixels.

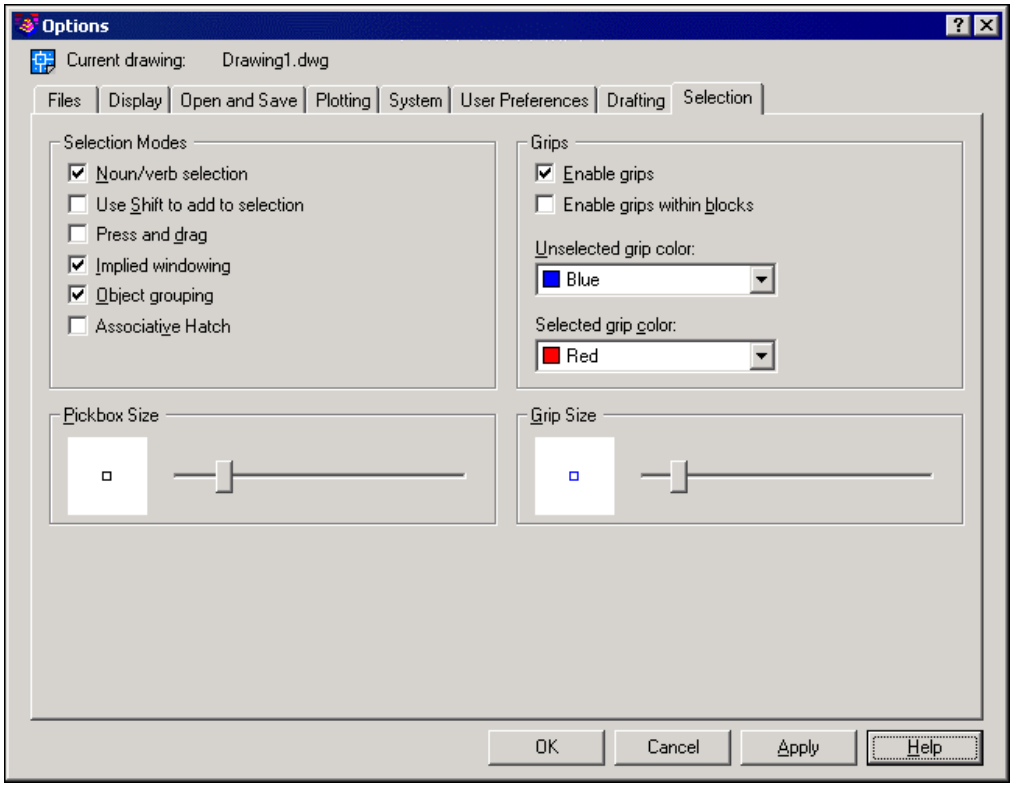
3 Under Tracking Settings, you control the settings that relate to tracking behavior.

- **Display Polar Tracking Vector**: Sets polar tracking behavior on or off. With polar tracking, you can draw lines along angles relative to a drawing command From or To point. Polar angles are 90-degree divisors, such as 45, 30, and 15 degrees.

- **Display AutoTrack Tooltip**: Controls the display of the AutoTrack tooltip. The tooltip is a text flag that displays the tracking coordinates. You can turn Object Snap Tracking on and off on the Object Snap tab in the Drafting Settings dialog box. You can also enable the AutoTrack tooltip by setting AUTOSNAP to 32.

4 Under Aperture Size, you set the display size for the Autosnap aperture. When Display AutoSnap Aperture Box is selected (or when APBOX is set to 1), the aperture box is displayed in the center of the crosshairs when you snap to an object. The size of the aperture determines how close to a snap point you can be before the magnet locks the aperture box to the snap point. The smaller the aperture, the closer you must be to the snap point to activate the magnet. Values range from 1 to 50 pixels. You can also set Aperture Size by using the APERTURE system variable.

Selection Tab



Under the Selection Tab, you control settings that relate to object selection methods.

1 Under Selection Modes, you determine the methods of selecting objects.

- **Noun/Verb Selection**: Allows you to select an object before starting a command. The command affects the previously selected object or objects. You can also set this option by using the PICKFIRST system variable.

- **You can use many editing and inquiry commands with noun/verb selection, including**: ALIGN, DVIEW, PROPERTIES, ARRAY, ERASE, ROTATE, BLOCK, EXPLODE, SCALE, CHANGE, LIST, STRETCH, CHPROP, MIRROR, WBLOCK, COPY, and MOVE.

- **Use Shift to Add to Selection**: Adds or removes an object to the selection set when you press SHIFT and select an object. To clear a selection set quickly, draw a selection window in a blank area of the drawing. You can also set this option by using the PICKADD system variable.

- **Press and Drag**: Draws a selection window by selecting a point and dragging the pointing device to a second point. If this option is not selected, you can draw a selection window by selecting two separate points with the pointing device. You can also set this option by using the PICKDRAG system variable.

- **Implied Windowing**: Initiates the drawing of a selection window when you select a point outside an object. Drawing the selection window from left to right selects objects inside the window's boundaries. Drawing from right to left selects objects within and crossing the window's boundaries. You can also set this option by using the PICKAUTO system variable.

- **Object Grouping**: Selects all objects in a group when you select one object in that group. With GROUP you can create and name a set of objects for selection. You can also set this option by setting the PICKSTYLE system variable to 1.

- **Associative Hatch**: Determines which objects are selected when you select an associative hatch. If this option is selected, boundary objects are also selected when you select an associative hatch. You can also set this option by setting the PICKSTYLE system variable to 2.

2 Under Pickbox Size, you control the display size of the Carlson Survey pickbox. The pickbox is the object selection tool that appears in editing commands. The default size is set to 3 pixels; values range from 0 to 20. You can also set the Pickbox Size by using the PICKBOX system variable. If you use the command line to set Pickbox Size, values range from 0 to 32767.

3 Under Grips, you control the settings that relate to grips. Grips are small squares displayed on an object after it has been selected.

- **Enable Grips**: Controls whether grips are displayed on an object after you select it. You can edit an object with grips by selecting a grip and using the shortcut menu. Enabling grips in a drawing significantly affects performance. Clear this option to optimize performance.

- **Enable Grips within Blocks**: Controls how grips are displayed on a block after you select it. If this option is selected, AgStar displays all grips for each object in the block. If this option is cleared, the program displays one grip located at the insertion point of the block. You can edit an object with grips by selecting a grip and using the shortcut menu.

- **Unselected Grip Color**: Determines the color of an unselected grip. If you choose Other from the color list, Carlson Survey displays the Select Color dialog box. the program displays an unselected grip as the outline of a small square. You can also set Unselected Grip Color by using the GRIPCOLOR system variable.

- **Selected Grip Color**: Determines the color of a selected grip. If you choose Other from the color list, Carlson Survey displays the Select Color dialog box. The program displays a selected grip as a filled small square. You can also set Selected Grip Color by using the GRIPHOT system variable.

4 Under Grip Size, you control the display size of grips. The default size is set to 3 pixels; values range from 1 to 20. You can also set the Grip Size by using the GRIPSIZE system variable. If you use the command line to set Grip Size, values range from 1 to 255.

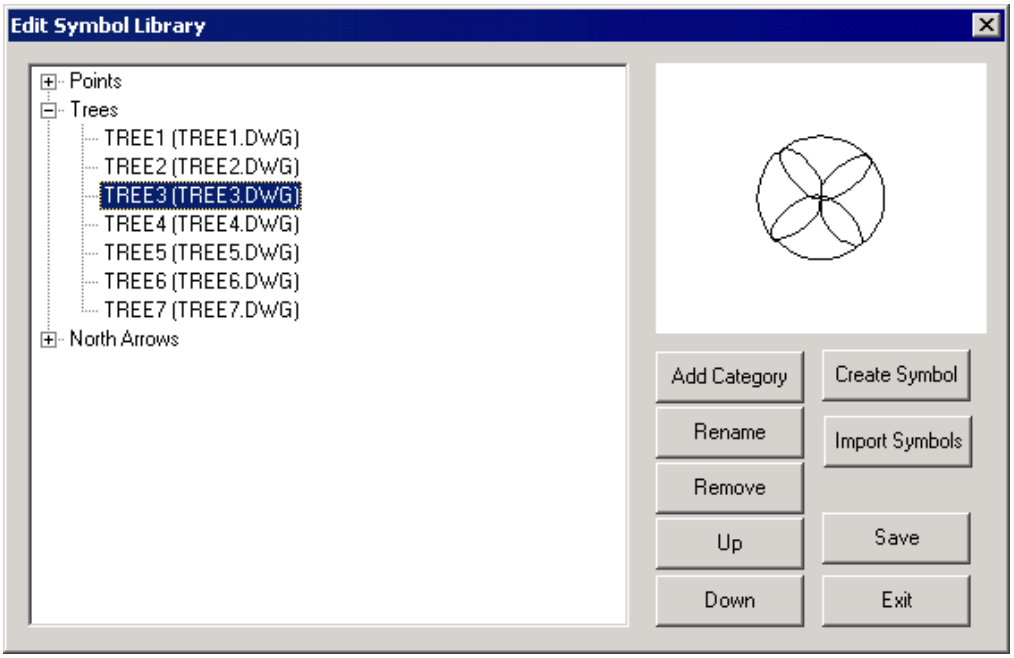
Menu Location: Settings

Prerequisite: None

Keyboard Command: PREFERENCES

Edit Symbol Library

This command allows you to customize the symbol library. The symbols are sorted alphabetically within each category, while categories are remaining in the order placed to allow the more frequently accessed categories be on top.



- **Add Category**: Categories are a way for grouping symbols by type for your own convenience in symbol selection. A new category is added by selecting this button. An edit field then appears in the tree view on the left and waits for you to enter the category name. Pressing the Enter key finishes the input.
- **Rename**: Select the category or symbol that you want to rename and press this button. By default, the symbol description is the same as file name.
- **Remove**: Select the category or symbol that you want to remove and press this button
- **Up**: If a symbol is selected, this moves the symbol up into the next category. If a category is selected, this moves the category up in the list.
- **Down**: If a symbol is selected, this moves the symbol down into the next category. If a category is selected, this moves the category down in the list.
- **Create Symbol**: Allows you to select drawing entities to create a new symbol. The symbol should be drawn at unit size (scale 1:1) because Carlson Survey will scale the symbol by the current drawing scale when the symbol is used.
- **Import Symbols**: Allows you to select existing drawing (.DWG) files to populate the selected category. If the files you select are not in the Carlson Survey SUP directory, the program will offer an option of copying them there.
- **Save**: Saves the symbol library list.
- **Exit**: Exit the dialog. If there are unsaved changes, you will be prompted to save.

Notes: The symbol library is stored in an ASCII file named symbols.dta in the Carlson Survey \USER directory.

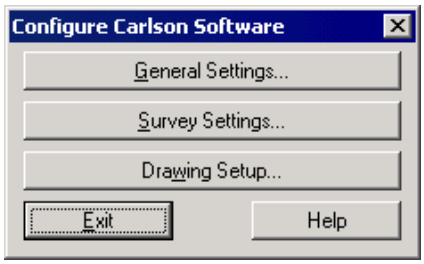
Menu Location: Settings
Prerequisite: None
Keyboard Command: EDITPTSYM

Configure

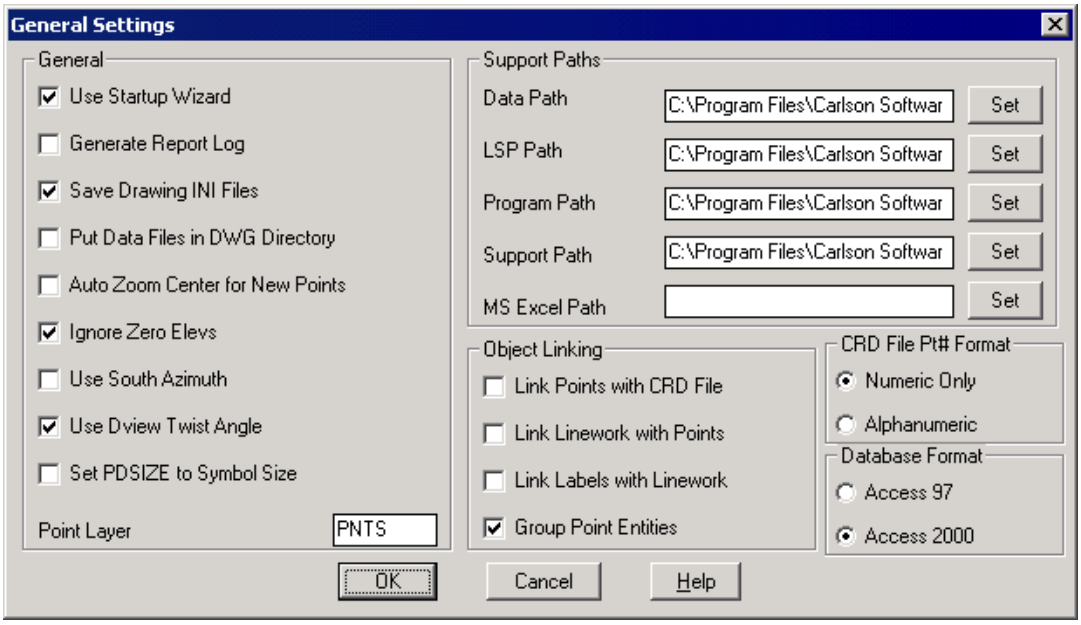
Function

This command allows you to set the default settings that are used each time you start a new drawing or load an existing drawing. These settings are stored in ∗.ini files in the Carlson Survey directory. Configure restores the current drawing settings to these default settings.

1 In the Configure dialog box you choose between General Settings, Survey Settings, and Drawing Setup.



2 In the General Settings dialog box you can set options for Carlson Survey.



A Under General, you can choose options relevant to points, angles, and start up.

- **Use Startup Wizard**: This option controls whether the wizard appears when you create a new drawing.
- **Generate Report Log**: This option allows output from several commands to be accumulated in a report buffer. Any report that is displayed in the standard report viewer is also added to the report log. While activated, the report log resides in the lower left corner of the desktop as a minimized title bar displaying the number of lines in the report buffer. To view the report log, pick the maximize icon on this title bar. You can edit the report log, save it to a file, or print it. To quickly turn the report log on and off, you can type REPORT at the command prompt, which toggles the report log on/off.
- **Save Drawing INI Files**: This option creates an .ini file with the same name as the .dwg file to store the project data files for the drawing.
- **Put Data Files in DWG Directory**: This option sets the Data Path to the directory of the drawing. The Data Path is the default directory for data files such as the coordinate file (.crd).
- **Auto Zoom Center for New Points**: This option will zoom center on new points.

- **Ignore Zero Elevs**: This option causes entities with zero elevations to be excluded from calculations, etc.
- **Use South Azimuth**: This option allows you to use a south azimuth for calculations.
- **Use Dview Twist Angle**: This option keeps text horizontal to a twist screen view.
- **Set PDSIZE to Symbol Size**: When checked, the system variable PDSIZE will be set to the same size as the symbol size that you set in Drawing Setup. PDSIZE controls the display size of AutoCAD point entities. Normally AutoCAD point entities are displayed as a dot, and the size does not apply. You may modify the point display type by changing the system variable PDMODE. For example, if you set PDMODE to 64, point entities are displayed as a square regardless of the Carlson Survey symbol type used.
- **Point Layer**: You can assign a layer name for points.

B Under Support Paths, you must determine paths for file allocation and retrieval.

- **Data Path**: You determine where data files (∗.crd, ∗.grd, etc.) are stored.
- **LSP Path**: You determine where command files (∗.dcl, ∗.arx, etc.) are stored. Warning: Changing this path can render AgStar non-functional
- **Program Path**: You determine where ∗.exe files are located. Warning: Changing this path can render Carlson Survey non-functional
- **Support Path**: You determine where auxiliary ∗.dwg files (point symbols, linetypes, etc.) are stored
- **MS Excel Path**: You determine the path for the ∗.exe file for MS Excel is located.

C Under Object Linking, you can set reactors to the drawing entities.

- **Link Points with CRD File**: This option attaches a reactor to the point entities so that any change to the entities such as MOVE or ROTATE will update the coordinates in the coordinate file.
- **Link Linework with Points**: This option attaches reactors to line and polyline entities that are drawn by point number so that moving the points automatically moves the linework.
- **Link Labels with Linework**: This option applies to bearing/distance annotation. This link updates the annotation when the linework is modified.
- **Group Point Entities**: This option joins the three entities of a point (attribute block, symbol, node). For each point, selecting any one of these entities selects all three entities for the point.

D Under CRD File Pt# Format, you can set the point number format for coordinate files as either numeric only or alphanumeric.

- **Numeric Only**: This options allows for numeric point numbers only.
- **Alphanumeric**: This option allows point numbers to contain characters, for example 5A, 5B, and 5C.

E Under Database Format you choose between Microsoft Access 97 or 2000 format. This only applies to new .MDB files created by Carlson Survey

F Under Report Print Specifications, you determine formatting for the Standard Report Viewer.

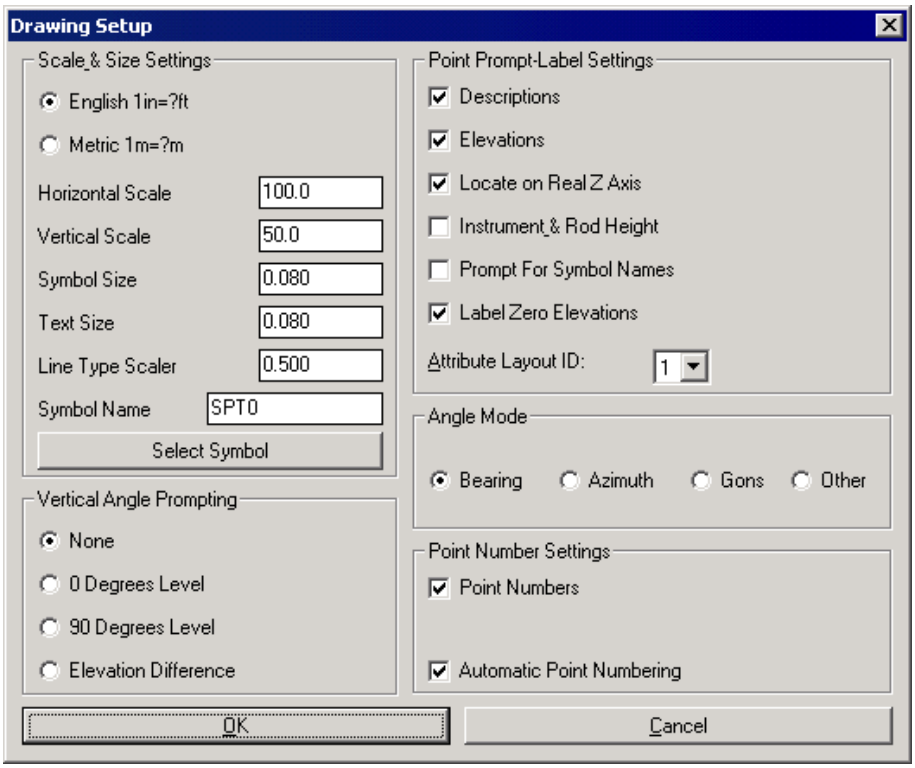
- **Use Degree Symbol in Reports**: This option lets you choose between using the degree circle symbol and a "d" for degree angles in the report. The "d" can be used for printers that don't handle the degree circle symbol.
- **Use Print Settings**: This option allows you to use settings on the Plot Settings Tab under Plot in the File menu.
- **Use Page#**: This option adds a page number to the bottom of each page in the report.
- **Text Size**: This option specifies text size for the reports.
- **Lines/Page**: This option specifies the number of lines per page of the report viewer.
- **Left Margin**: This option specifies the left margin of the report.
- **Top Margin**: This option specifies the top margin of the report.
- **Bottom Margin**: This option specifies the bottom margin of the report.

G Under Title Block, you set the height and width of the title block.

H Under Contour Options, you set the smoothing and offset parameters.

- **Smooth Contours**: This option automatically applies a smoothing factor to the calculated contour line.
- **Low/High slider**: This slider bar determines the amount of smoothing to be applied.
- **Reduce Vertices**: This option automatically applies a reduction to the number of vertices created in the calculating the contour lines.
- **Offset Distance**: This option determines offset distance of the contour lines.

3 In the Drawing Setup dialog box you have options for setting drawing parameters, including the plotting scale, size of symbols, label annotation size, and the drawing mode.



A Under Scale and Size Settings, you can determine scale and size of drawing entities.

- **English 1in=?ft**: This option tells the program which unit mode to use. This affects the prompting and reports. When you are working on a drawing in English units, one unit equals one foot.
- **Metric 1m=?m**: This option sets the metric scale to meters only.
- **Horizontal Scale**: This option allows you to set the horizontal scale of the drawing. For example, if the horizontal scale is set to 50, then 1" = 50' in your drawing.
- **Vertical Scale**: This option allows you to set the vertical scale of the drawing.
- **Symbol Size**: . This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Symbol Plot Size is not entered in Drawing Units
- **Text Size**: This value is a scaler that represents the size on the plot. The Drawing Units are determined by multiplying the scaler by the horizontal scale. In English mode the scaler represents the plotted size in inches. In Metric mode, this value is the plotted size in centimeters. The Text Plot Size is not entered in Drawing Units.
- **Line Type Scaler**: This option sets the linetype scale by multiplying this scaler by the horizontal scale.
- **Symbol Name**: This option allows you to set the default symbol name for points.

- **Select Symbol**: Click this button to graphically select the default symbol.

B Under Point Prompt-Label Settings, select the options that determine how the points are to be labeled and how you will be prompted for point entry.

- **Descriptions**: Determines whether you are prompted for a point description when creating points and whether the point descriptions are labeled in the point block.
- **Elevations**: Sets prompting and labeling for point elevations.
- **Locate on Real Z Axis**: Switches between locating points at zero elevation and at the actual stored elevations.
- **Instrument & Rod Height**: Turns on prompting for instrument and rod heights when creating points.
- **Prompt for Symbol Name**: When checked, the program will prompt for a symbol name as each point is drawn. Otherwise, the default symbol name set in this dialog box will be used.
- **Attribute Layout ID**: Controls the location of the point number, elevation, and description. These attribute layouts are defined in drawings that are stored in the AgStar Support directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then edit the associated SRVPNO drawing.

C Under Angle Mode, you determine how angles are entered and displayed.

- **Bearing**: This option sets reporting to bearing mode for any of the Inquiry commands.
- **Azimuth**: This option sets reporting to north based azimuth mode for any of the Inquiry commands.
- **Gon**: This option sets reporting to gon mode for any of the Inquiry commands.
- **Other**: Allows you to set a custom angle mode by using the Units Control command (described later in this chapter).

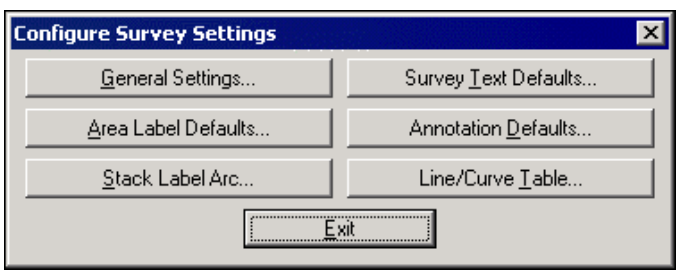
D Under Vertical Angle Mode select an option to determine how the vertical angle is calculated. Vertical Angle Prompt applies to creating points with commands such as Traverse.

- **None**: The vertical angle will not be used to calculate point elevations.
- **0 Degrees Level**: The vertical angle is used to calculate elevation and horizontal distance.
- **90 Degrees Level**: The zenith angle is used to calculate elevation and horizontal distance.
- **Elevation Difference**: Use the elevation difference to calculate the elevation.

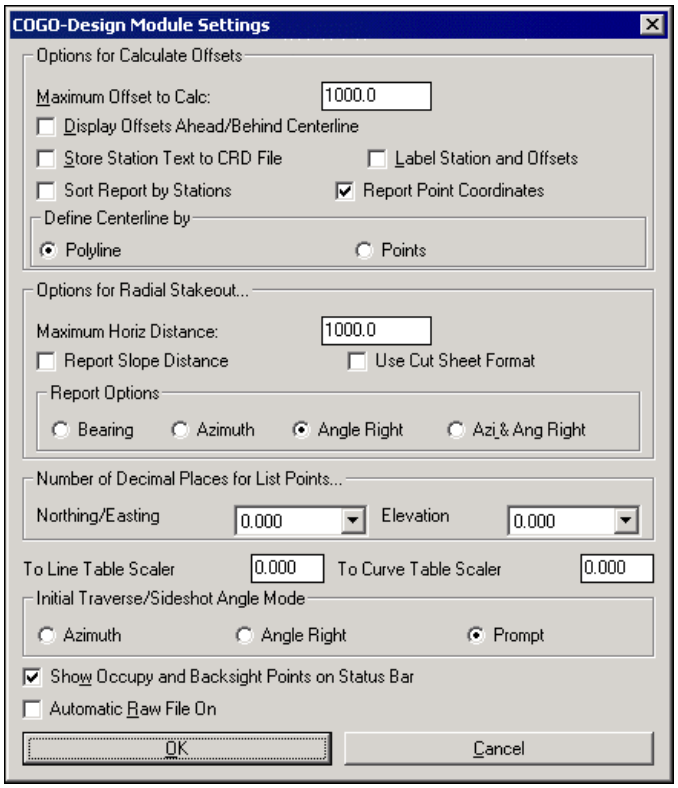
E Under Point Number Settings, select the options that determine whether you will be prompted for point numbers by the commands that locate points.

- **Point Numbers**: When this toggle is checked on, points that are inserted by AgStar commands are shown with a point number, and a coordinate is stored in the current coordinate (.crd) file. When this toggle is off, points are shown with no point number plotted and no coordinate is stored in the current coordinate (.CRD) file.
- **Automatic Point Numbering**: If this toggle is checked on, commands that locate a point will automatically insert a point number for each point drawn on the screen. If Automatic Point Numbering toggle is off, commands that locate a point will prompt for a point number.

4 In the Configure survey Settings dialog box, you set defaults for coordinate geometry calculations and design work.



5 In the General Settings dialog box under COGO-Design, you can set defaults for offsets and stakeout.



A Under Options for Calculate Offsets, you must set defaults for offsets.

- **Maximum Offset to Calc**: This option allows you to set the maximum distance an offset will be calculated from an origin.
- **Display Offsets Ahead/Behind Centerline**: This option allows you to display the calculated offsets apart from the centerline.
- **Store Station Text to CRD File**: This option allows you to save station text in the coordinate file with the points.
- **Label Station and Offsets**: This option allows you to label the generated stations and offsets.
- **Sort Report by Stations**: This option allows you to sort the output report by generated stations.
- **Report Point Coordinates**: This option allows you to report coordinates for all points generated.

B Under Define Centerline by, you must determine if centerlines are picked as polylines or as points.

C Under Options for Radial Stakeout, you must set defaults for the report, including angle format.

- **Maximum Horiz Distance**: This option allows you to define the maximum horizontal distance.
- **Report Slope Distance**: This option allows you to include the slope distance in the report.
- **Use Cut Sheet Format**: This option allows you to use the cut sheet format in the report.
- **Azimuth**: This option reports azimuth for radial stakeout.
- **Angle Right**: This option reports angle right for radial stakeout.
- **Both**: This option reports both azimuth and angle right for radial stakeout.

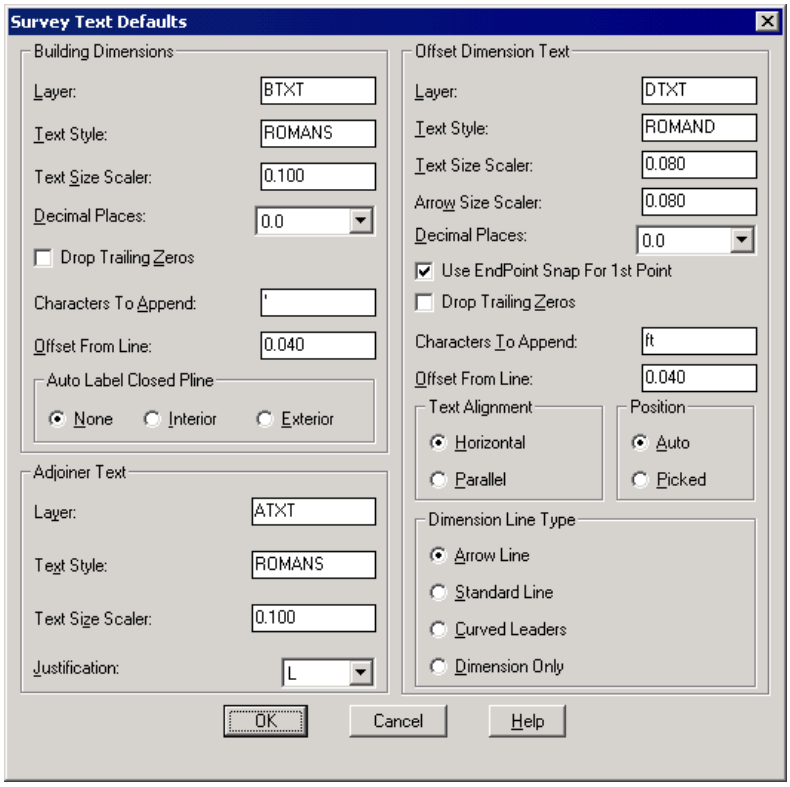
D Under Number of Decimal Places for List Points, you can set the precision for both Northing/Easting and elevation. You also set scale factors for line and curve tables.

E Under Initial Traverse/Sideshot Angle Mode you can determine if angles are represented as azimuth or angle right. Alternatively, you can set the default to prompt you each time.

F Show Occupy and Backsight Points on Status Bar: When checked, the current occupied and backsight point numbers are visible in the program status bar.

G Automatic Raw File On: When checked, a raw (.RW5) file is automatically created during any of the following commands: Locate Point, Traverse, Sideshot, and Inverse.

8 In the Survey Text Defaults dialog box, you can set defaults for building, dimension, and adjoiner text.



A Under Building Dimensions, you set text specification for building dimensions.

- **Layer**: This option allows you to set the layer for the building text.
- **Text Style**: This option allows you to set the text style for the building text.
- **Text Size Scaler**: This option allows you to set the text scaler to determine text size.
- **Decimal Places**: This option allows you to set the precision for the building dimensions.
- **Drop Trailing Zeroes**: This option allows you to truncate trailing zeroes from dimensions.
- **Characters To Append**: This options allows you to set characters to add to reported dimensions.
- **Offset From Line**: This option sets the offset distance from the line to the dimension text.
- **Auto Label Closed Pline**: This option specifies whether a closed polyline is labeled on the interior or exterior.

B Under Offset Dimension Text, you can set text specifications for offset dimensions.

- **Layer**: This option allows you to set the layer for the offset text.
- **Text Style**: This option allows you to set the text style for the offset text.
- **Text Size Scaler**: This option allows you to set the text scaler to determine text size.
- **Arrow Size Scaler**: This option allows you to set the arrow scaler to determine arrowhead size.
- **Decimal Places**: This option allows you to set the precision for the offset dimensions.
- **Drop Trailing Zeroes**: This option allows you to truncate trailing zeroes from dimensions.
- **Characters To Append**: This options allows you to set characters to add to reported dimensions.

- **Offset From Line**: This option sets the offset distance from the line to the dimension text.
- **Text Alignment**: This options allows you to align text either parallel to the line, or horizontally in the drawing.
- **Position**: This options allows you to determine if you will pick the location of the text, or if it is automatically positioned in the drawing.

C Under Adjoiner Text you can set text specifications for adjoiner text.

- **Layer**: This option allows you to set the layer for the adjoiner text.
- **Text Style**: This option allows you to set the text style for the adjoiner text.
- **Text Size Scaler**: This option allows you to set the text scaler to determine text size.
- **Justification**: This option allows you to set the text justification.

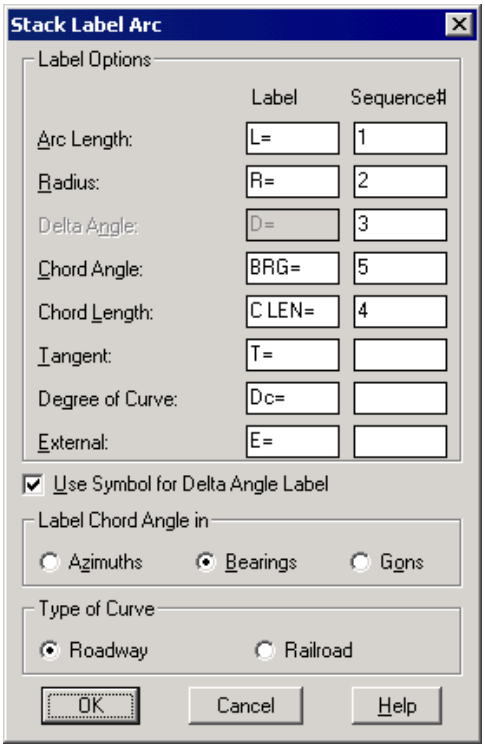
D Under Dimension Line Type, you can determine the line style to use for dimensions.

- **Arrow Line**: This option draws a line with an arrowhead from the dimension text to the figure.
- **Standard Line**: This option draws a line with no arrowhead from the dimension text to the figure.
- **Curved Leaders**: This option draws a curved line with an arrowhead from the dimension text to the figure.
- **Dimension Only**: This option draws the dimension text with no line.

7 In the Area Label Defaults dialog box you can set parameters for labeling area calculations. See the Area Label Defaults command in the Area chapter of this manual for a complete description of this dialog box.

8 In the Annotation Defaults dialog box you can set parameters for displaying annotation text. See the Annotate Defaults command in the Annotate chapter of this manual for a complete description of this dialog box.

9 In the Stack Label Arc dialog box, you can set parameters for the labeling of arcs.



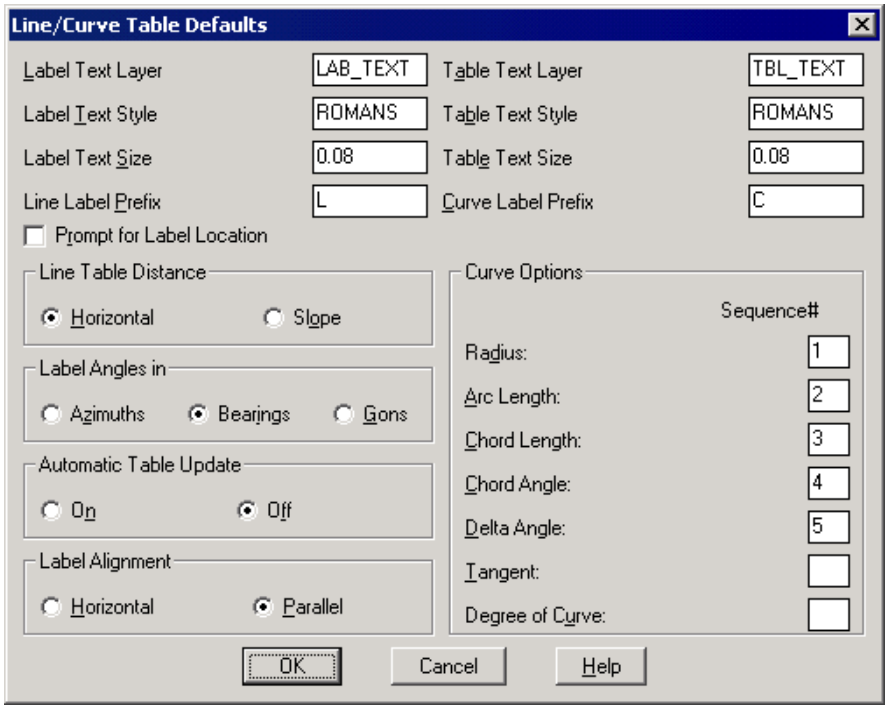
A Under Label Options, you can determine the label abbreviations and sequence in the label stack.

- **Options**: You must set the sequence and the label abbreviations for all the attributes of the arc label.
- **Use Symbol for Delta Angle Label**: This option inserts a symbol to denote the delta angle. If this option is unchecked, then you may specify the prefix for Delta Angle above.

B Under Label Chord Angle in, you can determine if the chords of the arc are labeled as azimuths, bearings, or gons.

C Under Type of Curve, you can specify if the arc is a roadway or railway curve.

10In the Line/Curve Table defaults dialog box, you can set parameters for producing line and curve tables.



A In the Line/Curve Table dialog box, you must assign text size and layers.

- **Label Text Layer**: This option sets the layer for the label text.
- **Label Text Style**: This option sets the text style for label text.
- **Label Text Size**: This option sets the text size for label text.
- **Line Label Prefix**: This option sets the label prefix for lines.
- **Table Text Layer**: This option sets the layer for the table text.
- **Table Text Style**: This option sets the text style for table text.
- **Table Text Size**: This option sets the text size for table text.
- **Curve Label Prefix**: This option sets the label prefix for curves.
- **Prompt for Label Location**: This option prompts you for label position.

B Under Line Table Distance, you can choose between slope or horizontal distance.

C Under Curve Options, you can determine sequence of the curve table.

D Under Label Angles in, you determine if angles are labeled in azimuths, bearings, or gons.

E Under Automatic Table Update, you determine if the table is automatically updated when lines or curves are modified.

F Under Label Alignment, you set the label as horizontal or parallel.

Menu Location: Settings

Prerequisites: None

Keyboard Command: CONFIG_SCAD

Survey Commands 7

Configure Survey

This command sets the equipment type, communication parameters and other AgStar options. Make sure the *Equipment Type* box shows the correct GPS or Total Station equipment that you'll be using. The down triangle button to the right of this box brings up a list of the equipment types to choose from. The eight buttons in *Configure* bring up the dialog boxes which are used to change AgStar's default settings. Explanations for each are shown below.

General Settings

If you are using a total station, *Rod Height* is the distance from the prism to the ground. For GPS, *Rod Height* is the distance from the center of the GPS antenna to the ground.

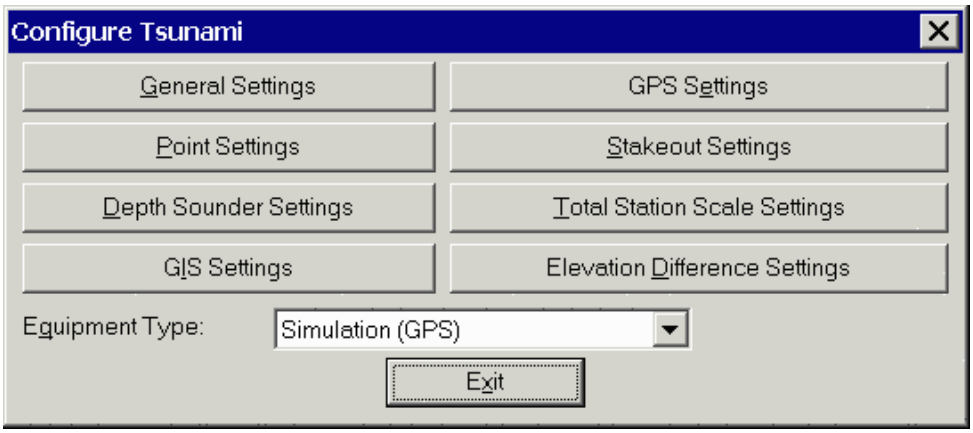
The *Show AgStar Startup Icon* controls whether the AgStar Startup Icon is displayed in the lower right of the screen. This startup icon brings up the AgStar function menu for launching AgStar commands without having to pick them from the pull-down menu.

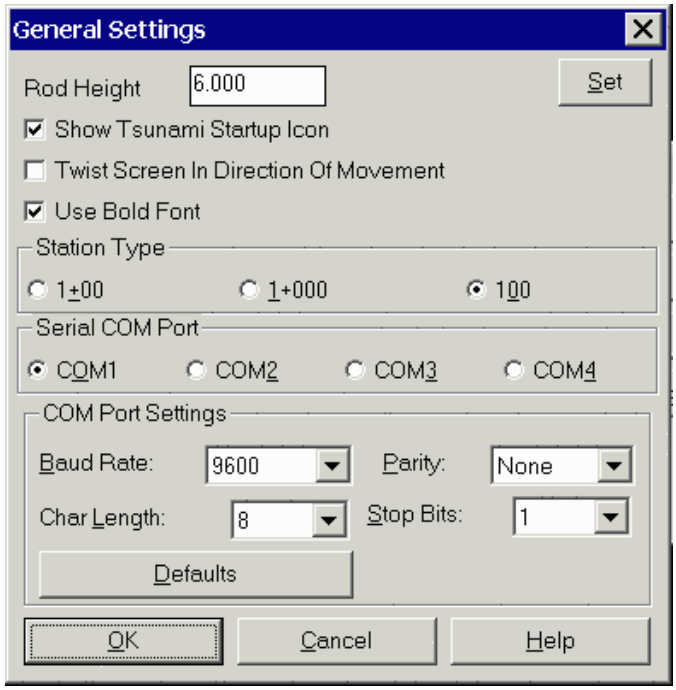
The *Use Bold Font* toggles between using standard or bold font for the AgStar dialogs.

The *Twist Screen In Direction Of Movement* will rotate the drawing view so that your current direction of movement is facing straight up in the view. This rotate is for the view only and does not change the coordinates. This option only applies to GPS and robotic total stations in commands that show the arrow icon such as Track Position.

The *Station Type* chooses the format of centerline station labels. Typically 1+00 is used for feet units, 1+000 is used for metric and 100 has no plus symbol in the number.

Serial COM Port - The GPS receiver or total station attaches to your AgStar computer us





ing a serial cable. This cable is plugged into a serial COM port on your computer called 1, 2, 3 or 4 . Check the circle denoting the COM Port to be used.

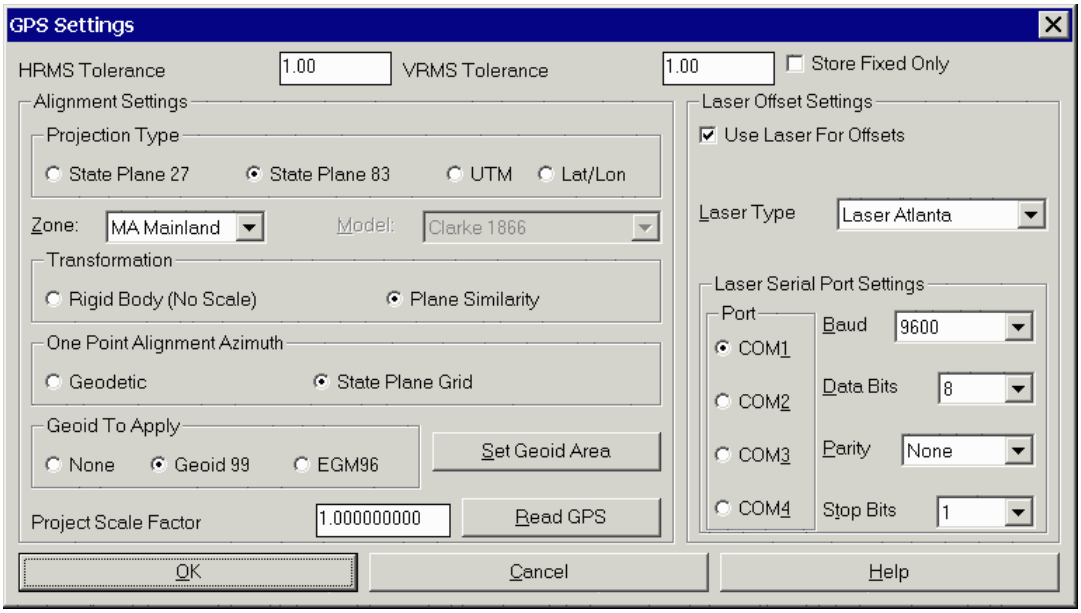
The *Baud Rate*, *Parity*, *Char Length* and *Stop Bits* are the serial port communication parameters for the AgStar computer. These parameters need to match the parameters on the instrument that you are using. The Defaults button will set these communication parameters to the standard parameters for the current equipment type.

GPS Settings

The *RMS Tolerance* checks the RMS values when reading GPS positions. The RMS is the accuracy value reported by the GPS receiver. There are separate settings for the horizontal and vertical RMS values. The RMS (root meaned squared) value means that the reported coordinate is within +/- the RMS value of the true coordinate to a certain confidence level. The confidence level depends on the GPS receiver. Typically it is a 98% confidence. If either RMS value exceeds the user-defined tolerance while storing points, AgStar will default to "No" when it asks if you want to store the point. You are required to choose yes to override the tolerance check and store the point.

Suggestion: When GPS RTK systems loose lock and go "Float", both the horizontal and vertical RMS values typically jump up to sub-meter (1' or higher) values. In AgStar, one foot is the default for the *GPS RMS Tolerance* . Some operators set the *GPS RMS Tolerance* low to 0.2 to check for high RMS values while still "Fixed".

Store Fixed Only - The position of the GPS rover is considered either "Autonomous", "Float" or "Fixed" based on the solution status from the GPS base corrections. When you are storing points and the *Store Fixed Only* box is checked, AgStar will only store points if your position is "Fixed". We suggest you leave this box checked. It ensures that you do not record inaccurate points.



Suggestion: When walking in light to heavy canopy, the rover might remain "Float" and display RMS accuracies of over a foot, sub-meter or more. Setting your *GPS RMS Tolerance* high and turning off *Store Fixed Only* will allow storing wetland and LOD (limits of disturbance) points under canopy that require only sub-meter tolerances. (USCG beacon DGPS sub-meter RTK GPS will always use these settings.)

Projection Type defines the datum coordinate system to be used for converting the latitude/longitude from the GPS receiver into cartesian coordinates. For the United States two separate horizontal control systems have been developed by the Federal Government: State Plane 1927 and State Plane 1983. For international use the UTM (Universal Transverse and Mecator System) should be selected. The Lat/Lon option will convert the latitude/longitude from degrees minutes seconds format into decimal degrees. This option is useful when working in a decimal degrees lat/lon coordinate system.

Zone - For State Plane projections, you must select the correct state zone that you are working in. For UTM, the Automatic Zone option will have the program automatically use the correct UTM zone for your location. Otherwise for UTM, you can manually set a specific UTM zone. This manual option applies to working on the border between zones and you want to force the program to always use one of those zones.

Important: Coordinates of surveyed points will be inaccurate if the Projection Type and Zone settings are wrong. If you have done survey work and then realize that they are set wrong, then your point coordinates are wrong, but your work is not wasted. AgStar records the latitude, longitude and height of every point in a ∗.RW5 file. You can input the correct projection zone settings later and reprocess your data using the Edit-Process Raw File command.

Model - For UTM, this option sets the ellipsoid constants for converting the lat/lon to UTM coordinates. The following is a list of the models:

Model Earth Radius(m) Flattening factor

Airy 1830 6377563.396 0.00334085064038

Modified Airy 6377340.189 0.00334085064038

Bessel 1841 6377397.155 0.00334277318217

Clarke 1866 ellipsoid 6378206.4 0.00339007530409

Clarke 1880 6378249.145 0.00340756137870

Everest(EA-India 1830) 6377276.345 0.00332444929666

Everest(EB - Brunei & E.Malaysia) 6377298.556 0.00332444929666

Everest(ED - W.Malaysia & Singapore) 6377304.063 0.00332444929666

International 1924 6378388.0 0.00336700336700

Helmert 1906 6378200.0 0.00335232986926

Hough 1960 6378270.0 0.00336700336700

Geodetic Reference System 1980 6378137.0 0.00335281068118
South American 1969 and Australian National 6378160.0 0.00335289186924

World Geodetic System 1972 6378135.0 0.00335277945417

World Geodetic System 1984 6378137.0 0.00335281066475

Transformation - The transformation in the Align Local Coordinates command can either be by plane similarity or rigid body methods. Both methods use a best-fit least squares transformation. The difference is that the rigid body method does a transformation with a translation and rotation and without a scale. The plane similarity does a rotation, translation and scale. This option only applies when two or more points are used in Align Local Coordinates.

One Pt Align Azimuth - This option applies to the rotation when using one point in Align Local Coordinates. For this alignment method, the state plane coordinate is translated to the local coordinate. Then the rotation can use either the state plane grid or the geodetic as north. No scale is applied in this transformation. The state plane and geodetic true north diverge slightly in the east and west edges of the state plane zone. This option allows you to choose which north to use.

Geoid To Apply - This option will account for the geoid undulation in determining the orthometric elevation of the measurement. The definition of the geoid model as currently adopted by the National Geodetic Survey is the equipotential surface of the Earth's gravity field which best fits, in a least squares sense, global mean sea level. Orthometric elevation measurements are used in survey calculations. In order to convert ellipsoid heights (He) as measured by GPS into orthometric elvations (Eo), you must provide for a correction between the GPS-measured ellipsoid (reference ellipsoid) and a constant leval gravitational surface, the geoid. This correction is the geoid undulation (Ug). The formula is He=Eo + Ug.

The Geoid models are essentially large elevation difference models in grid format. AgStar has two geoid models available. Geoid99 covers the United States at 1 minute grid intervals. EGM96 covers the entire globe at 15 minute intervals. These Geoid models are huge and take a lot of disk space and memory. The Geoid model files are not installed automatically and instead need to be installed by going to the Geoid folder on the AgStar installation CD. Once installed onto AgStar, you then need to specify your location by lat/lon so that the program only needs to load a local portion of the Geoid model. To set your local Geoid area, pick the *Set Geoid Area* button. Setting the Geoid area will carve out a Geoid model around the specified lat/lon covering a square area of 2 degrees by 2 degrees which is about 100 miles by 100 miles.

AgStar applies the Geoid model by subtracting the Geoid undulation from the GPS elevation. The resulting elevation is then used and displayed. In the Monitor function, the Geoid undulation is displayed.

In practice, the Geoid model is most applicable to two types of alignment scenarios. One of these types is when setting up the base over a known point and having no alignment control points. The other is when there is one alignment control point. When using multiple alignment control points, the Geoid model is not as important because AgStar can model the elevation

difference which can generally pick up the local Geoid undulation.

Project Scale Factor - After converting the LAT/LONG from the GPS to the state plane coordinates and applying the Align Local Coordinates, the Project Scale Factor is applied as the final adjustment to the coordinates. This adjustment is used on the X,Y and not the Z. The Project Scale Factor is applied by dividing the distance between the coordinate and a base point by the Project Scale Factor. The coordinate is then set by starting from the base point and moving in the direction to the coordinate for the adjusted distance. The base point is the first point in Align Local Coordinates. If there are no points specified in Align Local Coordinates, then 0,0 is used as base point. The Project Scale Factor can be entered directly or calculated using the grid factor and elevation for the current position. When using the current position, the program will read the LAT/LONG from the GPS receiver. The scale factor is then calculated as: (State Plane Grid Factor - (Elevation/Earth Radius)).

Laser Offset Settings - There is an option to use a laser for reading the distance andr angle for offset points. When this option is enabled, you can choose the laser equipment type and communication parameters. The serial port for

the laser must be different than the GPS which requires at least two serial ports on the computer. When using a laser for offsets, the program will read the current position from the GPS and then read the laser for the distance and angle to the point. This combination allows you to calculate points that cannot be directly reached by the GPS. There are two methods in the Point Store command to use the laser when this option is enabled. The Point Store dialog will have a new Laser button which will bring up another dialog that allows you to take multiple shots from the laser. The other method is to click on the Offset toggle in the Point Store dialog. Then when you do the Read function, the program will read the GPS position and then pop-up a dialog for taking one offset shot.

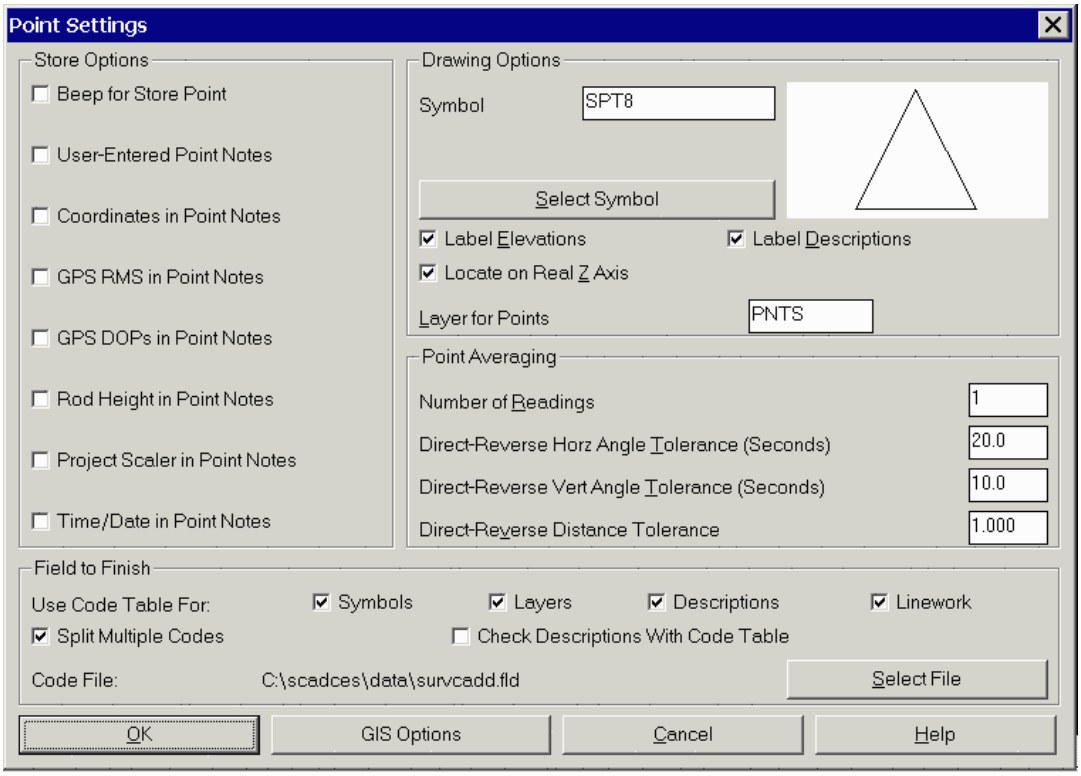
Point Settings

Beep for Store Point - This option will make a triple beep to indicate when a point is stored in the coordinate file.

User-Entered Point Notes - Point Notes are additional descriptions that can be stored with a point. A regular point consists of a point number, northing, easting, elevation and 32 character description. These points are stored in a .CRD file. Point Notes are a way to add an unlimited number of lines of text to a point number. With Point Notes ON in the Store Point command, the program will prompt for notes after collecting a point. The notes are stored in a file that uses the name of the coordinate file with a .NOT extension. For example, a coordinate file called JOB5.CRD would have a note file called JOB5.NOT.

Coordinates in Point Notes - When storing a point, this option will store the point number, northing, easting, elevation and description in the point notes as well. This may be used as a backup or reference to coordinate data as it was originally stored.

GPS RMS in Point Notes - When storing a point, this option will store the horizontal and vertical RMS values in the note field for the point. This offers a good check on the quality of the shot.



GPS DOPs in Point Notes - When storing a point, this option will store the DOP (dilution of precision) values as reported from the GPS receiver.

Rod Height in Point Notes - When storing a point, this option will store the rod height value in the note field for the point.

Project Scaler in Point Notes - When storing a point, this option will store the project scale factor in the note field for the point.

Time/Date in Point Notes - This option will store the time and date that the point was stored in the note file. AgStar will read the time from the computer.

Drawing Options control how points are drawn by default. It controls the layer, symbol number and whether points will be drawn with descriptions and elevations. AgStar's Field to Finish code table can override these defaults.

The symbol used for default points is displayed. You can choose another symbol by changing the *Symbol* name or by selecting one from the table that the *Select Symbol* button brings up. Default point settings are used for points whose descriptions don't correspond to any category

on the Field to Finish code table.

Label Descriptions and *Label Elevations* Control whether these two items of information appear on your drawing next to each point.

Locate on Real Z Axis will record points with their true elevations. If this setting is off, all points recorded will have an elevation of zero.

Layer for Points indicates the layer where all default points will be drawn. For points using a code on the code table, the code table will determine their layer.

Number of Readings specifies how many times AgStar will read from the instrument in the Read function of the *Point Store* command. This applies to both GPS and total stations. The readings will be averaged to find a more accurate position.

Direct-Reverse Tolerances are used with total stations to check the pairs of direct and reverse horizontal angles, vertical angles and distances. When these values are off by more than the tolerance, the program will display a warning.

Field to Finish is explained fully in the Field to Finish command definition. Basically it uses a code table which holds information on types of points (ie. Man Hole or Edge of Pavement). When the settings *Use Code Table...For Symbols, For Layers and For Descriptions* are selected, AgStar will look to the code table for how to draw points of a particular code description.

The file containing the active code table appears after *Code File:* You can change this with the button *Select File*.

The *Split Multiple Codes* option will draw multiple points from the same point when that point description has multiple codes. For example, a point with description "EP DR" will draw the point twice: once with the properties of code EP and a second time using code DR. When this option is off, the program will use the first code and draw the point once.

The *Check Descriptions With Code Table* option will display a warning beforing storing a point if that point description is not found in the code table. With this option off, the program will go ahead and store the point and the point will be drawn using the default point properties.

Stakeout Settings

Display GPS RMS in Stakeout causes AgStar to report the constantly updating horizontal RMS accuracy values while staking a point. The only disadvantage to having this option active is that it slows down a little the stakeout position update.

Draw Trail displays a line in the stakeout screen showing where you've been as you move towards the stakeout point. This option only applies to GPS.
Auto Zoom will zoom the drawing display in or out so that both your current position and stakeout target are visable on the screen.

Zero Horizontal Angle To Target will set the horizontal angle of the total station to zero in the direction towards the stakeout point. When stakeout is completed, the horizontal angle is set back to the original value. This option only applies to total stations.

Store Cutsheet/Stakeout Data in Note File will store stakeout data in the note file (.NOT) for the current coordinate file. At the end of staking out a point, there is an option to store the staked coordinates in the current coordinate file. This stakeout note file option allows you to store more stakeout data in addition to the staked coordinates. This

additional data includes the target coordinates and horizontal and vertical difference between the staked and target points. This stakeout note data can be used in reports with the List Points or CutSheet Report commands.

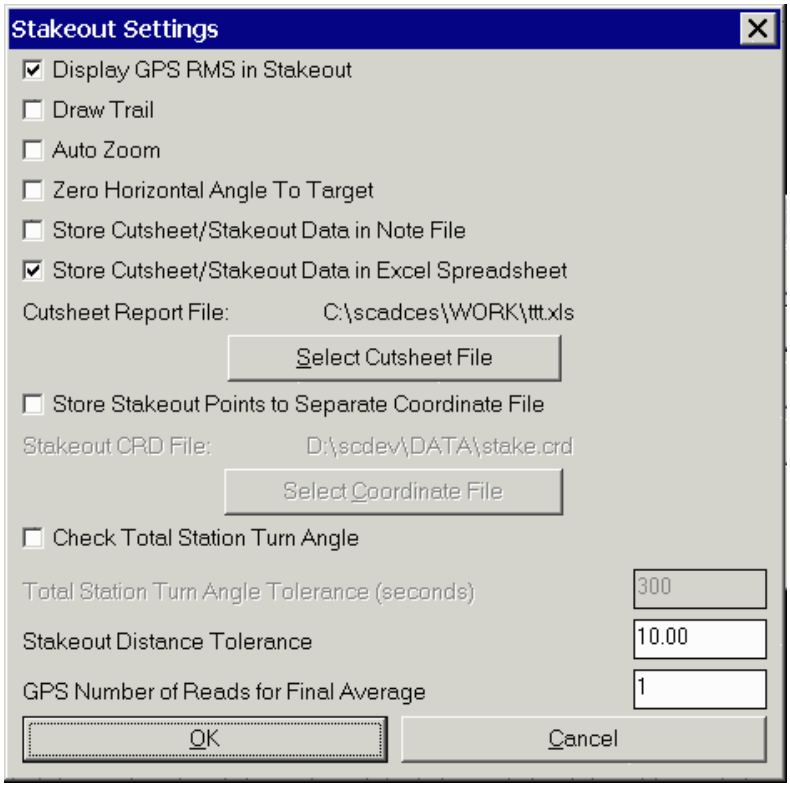
Store Cutsheet/Stakeout Data in Excel Spreadsheet will display a cutsheet report in an Excel spreadsheet. The spreadsheet will pop-up at the end of each point stakeout. The report can be saved in Excel format and processed by Excel.

Store Stakeout Points To Separate Coordinate File will store the staked points to a different coordinate file besides the current coordinate file. This allows you to use the same point number for the target and staked points. The staked point coordinate file can be specified by picking the Select Coordinate File button.

Check Total Station Turn Angle will compare the angle from the instrument and the angle to the target point. If this difference is greater than the specified tolerance, then AgStar will display a warning message.

Stakeout Tolerance controls the maximum difference between the target location and actual staked point. When the staked point is beyond the tolerance, AgStar displays a warning dialog.

GPS Number of Reads for Final Avg specifies how many times AgStar will read the GPS receiver position for the final staked point. These reading are averaged. Averaging several readings while occupying one point yields a more accurate result, but inevitably takes longer.



Total Station Scale Settings

These settings apply only to total stations.

The *Project Scale Factor* is multiplied by the measured distance from the total station when calculating the foresight point coordinates. A typical project scale factor for working in state plane coordinates is slightly less than one. Factors greater than 2.0 or less than 0.5 are not allowed. The Project Scale Factor can be entered directly or choose the Calculate button. The Calculate function takes a state plane coordinate and calculates the project scale factor as the state plane grid factor minus the elevation factor (Grid Factor - elevation/earth radius). The state plane coordinate is specified by a point number from the current coordinate file.

The *Calculate State Plane Scale Factor At Each Setup* option will calculate the scale factor for each shot as the combined grid and elevation factors (see above equation). The scale factor is calculated at both the occupied and

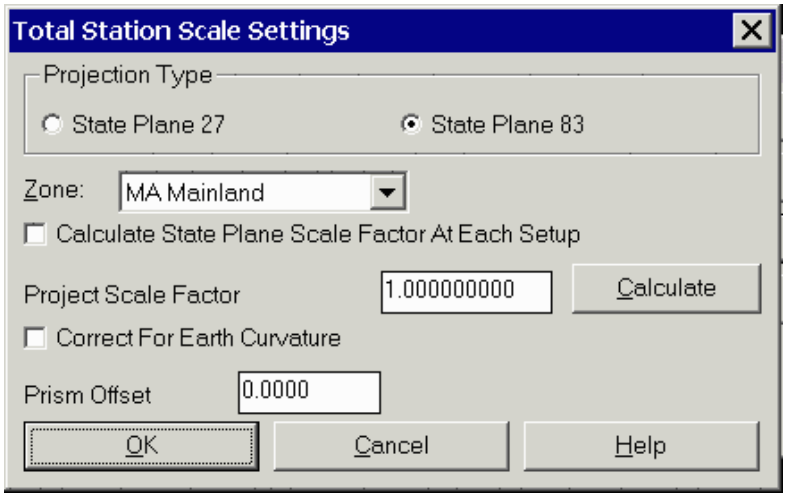
foresight points and then averaged. To use this option, you must be working in state plane coordinates and set the state plane zone in this dialog.

The *Correct For Earth Curvature* option adjusts the horizontal distance and vertical differece to the foresight point to account for the earth curvature.

Depth Sounder Settings

AgStar can use depth sounders in combination with GPS to collect points of underwater surfaces. AgStar supports depth sounders that output standard NMEA data, the Odom Digitrace model and the Hydrotrac model. For the Odom Digitrace, you also need to specify the depth unit mode that the instrument is set to.

The depth sounder must be connected to a separate serial port than the GPS. The *Baud Rate* between the computer serial port and the depth sounder is also specified here. The *Store Depth In Notes* option will record the water depth in the current note file (.NOT) when a point is stored to the coordinate file. The *Debug* number can be used when contacting technical support if the depth sounder is not communicating to AgStar



Elevation Difference Settings

These setting apply to the Elevation Difference command.

Grading Tolerance is the target difference between the actual elevation and the design surface.

AgStar can use an external Light Bar to indicate whether your current position is in cut, fill or on-grade. Currently AgStar supports light bars made by Apache and Mikrofyn. The Light Bar must be connected to a separate serial port than the GPS.

GIS Settings

A standard point is stored in the coordinate file with a maximum 32 character description. The GIS Settings allow you to store more data with each point.

The *Store Data In Note File* option will record additional fields for each point in the note file. The note file has the same name as the current coordinate file except with a .NOT instead of .CRD file extension. The fields that are recorded are defined by the GIS File (.GIS). This file defines a sequence of field names and prompts. For example, a GIS file for manholes could contain Location, Depth and Condition fields. Choose the Select File button to choose the GIS file to use. Or use the *Select GIS File Automatically by Point Description* to use different GIS files depending on the point description. With this option, the program will look for a GIS file with the same name as the point description. For example, if the point description is MH, then the GIS file will be MH.GIS. See the Define Note File Prompts command for more information.

The *Store Data Direct To Database* option will store additional fields for each point in a Microsoft Access database. The database to store the data is set in the Output File line. The Template File is a database that defines the fields to record. See the Define Template Database command for more information.

Equipment Commands

CSI GBX Pro

Geodimeter

Impulse Laser

Leica GPS System 500

Leica TC Series

Manual Total Station

Nikon Total Stations

Simulation (GPS)

Sokkia

Topcon Total Stations

Trimble

Navcom GPS Setup

Navcom Configuration Guide

CSI GBX Pro

Hardware Setup

1) Connect the receiver to the antenna by coaxial antenna cable if it is not already connected, and ensure that that the receiver has ample power.

2) Ensure that the antenna is tracking corrections from an MSK Radio Beacon. The easiest way to do this is to use the antenna's automatic frequency scanning when first powering on the receiver

a) To do this, enter the [SETUP] menu, and select the option [AUTO BX SEARCH]

-Note that the beacon automatically selected by this scan will be saved to the receiver's memory and used automatically in the future, until either the scan is executed again, or until a new beacon is specified manually. Thus, it is not necessary to scan each time the bacon is used, provided it is still operating in the same general area.

b) A scan can be performed again in the event that the beacon is lost to scan for the next nearest beacon.

3) Enter the [Setup] menu, then select [Options] then [NMEA ON/OFF]. This menu allows the enabling or disabling of various NMEA messages. The only ones which are necessary are the GGA, GSV and GSA messages. All others should be disabled.

Software Setup

4) In AgStar, no further setup is necessary to make use of the CSI GBX Pro. Simply use the other AgStar functions as normal. Note however, that the elevations reported by the CSI GBX Pro are MSL(Mean Sea Level).

Geodimeter

Geodimeter 600 For Remote Mode

Note: Firmware version 696-03.xx or higher is required on the instrument.

SET-UP

1. Connect the instrument to the battery pack. There is no need to connect the keyboard to the battery if it is going to be turned off, or attached to the unit. 2. Connect the prism to the top port of GeoRadio.

3. Connect the bottom port of the GeoRadio to AgStar. Then turn on the radio.

4. Turn on the Geodimeter. The Geodimeter starts with the screen for leveling the instrument. When the instrument is leveled press [ENT] key to continue to the next step. Now the instrument starts compensator calibration. You can

wait for calibration to finish or turn it off. To turn calibration off press on [F] 22, enter 0 for comp. This needs to be done when the instrument is turned on and before [ENT] is pressed.

5. Next Geodimeter will ask for different values for pressure, offset, etc. They can either be left like they are by pressing on [ENT] or they can be changed.

6. Press [F] 79, it is the End of Transfer character, which should be set to 4. 7. To set radio, and station channels, press [MNU], and enter 1 for "Set". After set press 5, which will give the user opportunity to change channel, station, and remote address.

NOTE: The channel, station and remote address on the Geodimeter should match the channel, station and remote address in AgStar.

8. To set the Geodimeter for remote mode, press on RPU, then 3 for remote and 1 for ok, you can answer [NO] to "Define Window?" If [ENT] is pressed, the instrument will ask "Aim to A Press Ent", for which the user have aim to upper/lower left boundary and press [ENT], for "Aim to B Press Ent", aim to the upper/lower right boundary and press [ENT]. For "Measure ref obj?" press [ENT] if you want a reference object, otherwise press [NO]. Than the instrument is going to say remove keyboard however the keyboard can stay on.

9. After Geodimeter display screen turns itself off, it's ready for AgStar.

AGSTAR 1. In Configure Survey, under equipment type there should be Geodimeter. In General Settings Baud Rate should be set to 9600.

2. After Configure Survey go to Total Station Setup and make sure GeoRadio is checked, and the channel, station and remote address is the same as it is in the total station.

NOTE: We recommend using channel 3.

3. If calibration box is checked the instrument will calibrate, to turn of calibration the box should be unmarked.

4. In setup there is also an option to turn on/off-tracking lights.

Geodimeter 600 For Direct Connection

1. Connect the instrument to the battery pack, and the control unit to AgStar.

2. Under AgStar go to Configure Survey and place Geodimeter in Equipment type.

3. Click on General Settings make sure that the baud rate is set to 9600.

4. Exit Configure Survey.

5. Go to Total Station Setup and check Connect to Station and click OK.

Now you are ready.

Impulse Laser

There are two types of Impulse Lasers from Laser Tech. One has a compass for horizontal angles. The other has only a laser range finder for distance. These are refered to in AgStar as Impulse 9600 and Impulse 4800 respectively. The default baud rate for the Impulse 9600 is 9600 and the baud rate for the Impulse 4800 is 4800. The baud rate is set in Configure Survey. To take a shot with the Impulse, press the button that is closest to you. With the Impulse 4800, AgStar will prompt for an azimuth after reading the distance.

If you do not get data with the Impulse 4800, make sure that the Impulse is configured for the correct format. To check the format, press the far key twice. The screen should show SYS at the top. Then press the close key. Keep pressing the middle key to view the different settings until you see IP 200 or Cr 400. AgStar needs Cr 400 format. So if the Impulse reads IP 200, press the near button once.

Leica GPS System 500

Setting Up a 500 Series Receiver

1.) Connect the antenna cable to the ANT Port on the front of the receiver, and to the antenna.

2.) If you are using the PacCrest radio module, screw it in place over Port 1 on the receiver and attach its antenna cable. Otherwise, connect any radio being used to Port 1, 2 or 3.

3.) If an external power source is being used, be sure to plug it into the PWR Port on the front of the

receiver.

4.) If external power is not being used, ensure that there are batteries in one or both of the batter slots on the bottom of the receiver.

5.) Plug the 9 pin serial connection cable into the serial port of the computer running AgStar and into the Terminal Port on the front of the receiver.

Configuring AgStar for Use With a 500 Series Receiver

1.) Select "Configure Survey" from the AgStar pull-down menu. This will open a new window with several buttons on it, as well as a pull down list labelled "Equipment Type." Select "Leica 500 Series" in the Equipment Type menu, then select "General Settings."

2.) Ensure that the COM port is set to the one that the serial cable is plugged into, and that the Baud Rate is 9600, the Char Length 8, the Stop Bits 1, and the Parity None. Close this menu and the Configure Survey menu.

3.) In the Survey pull-down menu, select "Equipment Setup." This will open another menu with several

selectable options and several buttons.

4.) Use the radio buttons on the top right to select whether the receiver will be a rover or a base station. Also be sure to select the antenna types being used from the pulldown menu at left.

5.) Enter the desired Satellite Elevation Cutoff in the text box above the column of buttons. All satellites with elevations less than this number will not be used in position calculation(receiver default is 15).

6.) Select the "Radio Settings" button. This will open another window with several selectable settings. Select the Port number the radio is attached to on the front of the receiver, the baud rate of the radio, number of radio stop bits and radio parity. These last three settings should be listed in the documentation for the radio being used. Also, select the desired format to use for sending and receiving messages from the bottomost option. Exit this menu.

7.) If the GPS receiver is being configured as a base station, select the "Configure Base Station" button from the Equipment Setup menu, and proceed with step 8. Otherwise, the receiver is ready for use.

8.) There will now be a menu with a few buttons to select a method of determining the base station's present location. The options are:

> Read From GPS- R ead one or more position readings from the GPS and use this position or the average of several poss itions for the base station corrections.

> Enter Lat/Lon- this option will bring up a menu to enter the exact Latitude, Longitude, and elevation of the receiver's position by hand.

> Enter State Plane Coord- This option will bring up a menu to enter the coordinates of the position of the base station according to the state plane coordinate system.

> Read from File- this option will read a coordinate set from a file already saved to the computer.

Select whichever method will be used, and enter any necessary data. The receiver is now configured and ready for use.

Other Buttons In the Setup Menu:

1.) Power Cycle Receiver- This will shut the receiver down and then power it up again. Used to clear the receiver's memory.

2.) Power off Receiver- Shuts the receiver down. Note that if this button is pressed, any settings changes made while in this menu will not be saved to the receiver.

3.) Send command to receiver- this will allow for sending messages to the receiver. The user must enter the message by hand. This feature is only intended for use in conjunction with the technical support provided with AgStar.

Troubleshooting the Leica 500 Series in AgStar

Several possible errors can occur in the course of using a 500 Series Leica receiver with AgStar. AgStar will use all its standard error messages to report usual types of error messages, such as an inabilty to communicate with

the satellites that are being tracked. In addition, the Leica 500 Series of receivers will have their own set of error messages unique to themselves. This type of error message is reported if there is an error during the transmission of various configuration messages to the receiver to set up the base station settings. Such messages will say "Set Port Message Rejected", or "Set Base Antenna Message Rejected" or "Set Antenna Height Message Rejected" or "Set RTK Message Rejected." Each indicates which particular facet of the configuration failed. If one of these messages is rejected, it is likely a momentary transmission error. If, on the other hand, several (or all) are rejected, it is possible there is a problem in the communication line between the computer and the receiver, which should be checked.

Leica TC Series

Remote Mode

1. Turn on Leica 2. Connect Leica to rover radio, and connect the radio to the larger battery.

3. Connect the base radio to AgStar, and the smaller battery.

4. Under Survey go to Configure Survey, and under equipment type put Leica TC

5. To make sure the baud rate matches, under Configure Survey menu click on General Settings and check if the baud rate is 19200. When Leica is turned on under Main Menu enter 5 for "Configuration", and 2 for "Communication Mode", then enter 1 for "Gsi parameters", and check if the baud rate is also set 19200.

6. Line Terminator in "Gsi paramiters" should be set to CR/LF

7. Press [F1] for "cont" when done.

8. When back in Communication Mode screen enter 5 for "RCS (Remote) ON/OFF" and make sure it's NOT set for remote mode.

9. In Main Menu press [F6] for "meas" when ready to measure.

10. In AgStar go to Total Station Setup and for Connection Mode check remote.

11. When done click on OK.

To put Leica in Tracking: On Gun press "FNC" then ATR+ and LOCk+

TCA 1800

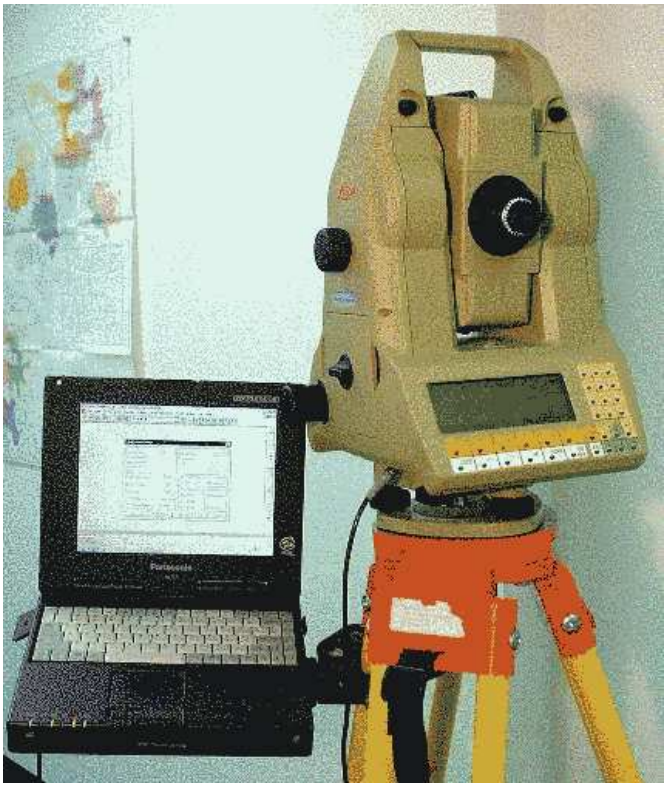
1. Turn on Leica

2. Connect Leica to AgStar

3. In AgStar go to Configure Survey, and under equipment type put Leica TC

4. To make sure the baud rate matches, under Configure Survey menu click on General Settings and check the baud rate. When Leica is turned on press [F3] for "conf", then enter 3. The baud rate can be changed by pressing [F6] for "list", when done enter [CONT]. In addtion to baud rate parity, char length, and stop bits should also match.

NOTE: Default in AgStar is not the same as default in Leica.



Manual Total Station

This method allows you to run AgStar in total station mode without being connected to equipment. The program will prompt you to enter the horizontal angle, zenith angle and slope distance. This method can be used for demonstration purposes or to work with total stations that cannot connect to AgStar. For these total stations, instead of the automatic connection, you can take a shot, read the instrument and then manually enter the data into AgStar.

As with other total stations, the first step is to run Total Station Setup to establish the occupied point, backsight and instrument/rod heights before running AgStar functions. Then in AgStar functions, when you pick the Read button, the program will bring up a dialog for entering the angles and distance. The angles should be entered in dd.mmss format (degrees.minutes seconds).

Navcom Configuration Guide

This guide will walk you through the setup process for your Navcom units. It covers individual unit setup as well as base/rover setup under the simplest possible configuration. If you want to customize the configuration, consult the reference manual

A) Preliminary setup steps

Perform the following preliminary steps to initialize your computer for communication with your Navcom units:

1) Startup the Carlson product you intend to use.

2) Select *Configure Survey*, from the *Survey* Menu.

3) Select *Navcom* from the *Equipment Type* dropdown

4) Click the *General Settings* button. Verify that the *Serial COM Port* is set to the port you intend to use to communicate with your Navcom unit. (usually COM1).

5) In the *COM Port Settings* box, click *Default*, and verify that baud rate=19200, parity=NONE, char length=8, and stop bits=1.

6) Click *OK*, and then click the *GPS Settings* button.

7) Under *General Settings*, set your HRMS and VRMS tolerance. For single-unit setup, these numbers should be at least 10. For base/rover configuration, they should be around 0.01.

8) Under *Projection Type*, select the coordinate plane you wish to use. For state plane, make sure you choose the proper zone.

9) Click *Ok* and then click *Exit*.

B) Single-unit setup (no corrections)

Before attempting a multi-unit setup, it is recommended that you first try setting up your Navcom unit to output an uncorrected position. The steps to do so are explained here:

1) Mount your GPS Antenna on a tripod in a place where its view of the sky is not obstructed.

2) If your antenna is separate from your receiver, connect your antenna to your receiver's antenna port. (This step can be skipped for the RT-3010S, and other all-in-one models)

3) Plug your receiver into a power supply, or insert fully charged batteries into the battery ports. (Not all units have battery ports).

4) Turn your receiver on by holding down the power button for a few seconds, or until the status lights flash on.

5) Use the serial port cable to connect your computer to port A of your Navcom unit. Make sure the port on your computer that you use corresponds to the one you chose during preliminary setup.

6) Under the *Survey* menu, choose *Equipment Setup*.

7) If a *Port Setup* window pops up, set *Control Port* to *Port A*, and *RTK Data Port* to *Radio Port*.

8) Setup is now complete. Steps that follow are optional.

9) Click the *Navigation Status* button. From here you can monitor the progress of your Navcom unit as it calculates its position.

10) It may take a few minutes for the unit to calculate its position, if the unit was reset, or newly turned on. When the calculation is complete, *Valid Navigation* will read *Yes*. When this occurs, the Navcom unit is ready for use. Click *Close*, then click *Cancel Without Saving*.

11) To monitor your position, choose *Monitor GPS Position* from the *Survey* menu, and you will see your current position. All AgStar GPS functions should now work.

C) Multi-unit setup (using corrections from a base)

1) Base Setup

a) Perform the preliminary and single-unit setup steps described above.

b) Attach the radio antenna to the radio port of your base unit.

c) Select *Equipment Setup* from the *Survey* menu.

d) Select a *Correction Type*. We recommend *NCT RTK*.

e) Click the *Configure Base* button.

f) When prompted to enter a position, enter the exact position of the base unit. Note that the accuracy of your rover's calculation depends on this position being completely accurate.

g) Enter a *Station ID* of 0. The *Station ID* is only used in RTCM mode.

h) Verify that *Station Type* now reads *Base*.

i) Base station setup is complete. Click *Save Settings and Exit*.

2) Switch the device you're plugged in to:

After configuring the base, unplug your serial cable from your base's port A, and plug it into your rover's port A.

Note: Whenever you switch the device you're plugged into be sure to close the GPS Setup window first.

3) Rover Setup

a) Perform the preliminary and single-unit setup steps described above.

b) Attach the radio antenna to the radio port of your rover unit.

c) Select *Equipment Setup* from the *Survey* menu.

d) Select a *Correction Type*. We recommend *NCT RTK*. Note that this selection must match the selection made during base setup.

e) Click the *Configure Rover* button.

f) Verify that *Station Type* now reads *Rover*.

g) Rover setup is complete. Steps that follow are optional.

h) You can verify that the rover is receiving correction by clicking the *Monitor Incoming Corrections*. The open window shows the time since each correction type was last received (delta time). In NCT RTK mode, the delta time of 5b, should stay around 1 second, and the delta time of message 5c should not go above 30 seconds. If these numbers are high, or if they read NEVER, try repeating the setup process or calling Carlson Software technical support.

i) Click *Save Settings and Exit*, and then choose *Monitor GPS Position* from the *Survey* menu. The *Status* display should eventually go to *Lock*.

Troubleshooting Base/Rover Configuration:

If you've configured a base to output corrections, and you're rover does not appear to be receiving the corrections, try each of the following in order:

1) Verify that your base and rover are both set to the same correction type.

2) Under *Configure Radio*, check that your rover is set to slave, and that your base is set to master.

3) Under *Configure Ports*, check that both your base and your rover's RTK Data Ports are set to the proper value (Usually *Radio Port*).

4) Under the *Edit Base Position*, check that your base is set to a valid position. Note that if the given position is too far away from the position the base is reading, the base will not send corrections.

5) If you're trying to use RTCM, make sure the base and rover have the same station ID's.

6) Try increase the RTK Max Age constraint.

7) Under *Navigation Status*, verify that the Navigation is valid on both units. If either unit does not have a valid position solution, correction wills not work.

8) Under *Monitor Corrections*, verify that the corrections you're using are arriving regularly. If they aren't you may need to reset both units.

9) Try configuring the base and rover again.

10) If all else fails, *Soft Reset* both units through the *Reset Unit* menu. After doing so, you will have to reconfigure the port settings of each device through the *Configure Ports* menu, and wait a few minutes for the devices to recalculate their position.

11) If none of these steps work, contact Carlson Software Technical Support.

Navcom GPS Setup

AgStar supports Navcom's NCT-2000D GPS message protocol, firmware versions 2.6 and later. If your Navcom unit has an earlier firmware version, contact Navcom for a free upgrade. AgStar has been tested extensively with Navcom models RT-3010S and RT-3020M.

From the Navcom GPS setup menu, or any of its submenus, the current device settings can be obtained by clicking the *Retrieve Settings* button. New settings can be saved by clicking the *Save Settings* or the *Save Settings and Exit* button. To cancel your changes, click *Cancel without Saving*.

By changing the *SV Elevation Mask*, you can prevent the Navcom Unit from using any satellite below a specified elevation angle (Range: 0-90). By changing the *PDOP Mask*, you can prevent the Navcom Unit from using any GPS solution with a PDOP above a specified value (Range:1-25).

By changing the *RTK Max Age*, you can prevent the Navcom Unit from using any RTK corrections older than a specified number of seconds. (Range: 0-1275, Multiple of 5).

By changing the *Base Station ID* on a base, you can provide your base with a unique identifying number so that rovers can specify which base they want to use for corrections. By changing this settingson a rover, you can specify which base unit you want to use. If 0 is specified, the rover will use any base station it can find. The base station ID only applies when using the RTCM correction format. (Range: 1-1023)

You can choose between 4 different *Correction Types*: NCT (Navcom Proprietary), CMR (Trimble's format), RTCM RTK (Messages 18-22), or RTCM DGPS (Message 1 and 9). When configured to BASE, changing the correction type changes the type being sent. When configured to ROVER, changing the correction type changes the type the unit is listening for. A ROVER will ignore all incoming correction messages except those of the type specified.

Configure Ports Submenu:

The *Control Port* should be configured to *Port A* or *Port B*, depending on which of the Navcom units' ports you are plugged into. (Note that the *Control Port* refers to the number of the port on the Navcom unit, NOT the number of the COM port on your computer) If the *Control Port* is configured improperly, you will not be able to communicate with your Navcom unit.

The *RTK Data Port* refers to the device port out of which RTK corrections will be sent. This value should be set to *Radio Port*, unless you want to set up a non-wireless Base/Rover connection through Port A or Port B. The *RTK Data Port* cannot be the same as the *Control Port*.

Configure Radio Submenu:

The *Radio ID* is the value used to identify a unit on a wireless network of Navcom units. Make sure that no other Navcom unit in your vicinity shares the same *Radio ID*. By default, the *Radio ID* is the same as your Navcom unit's serial number. This value can be changed, although there isn't usually any need to do so.

The *Local Radio Type* can be set to either *Master* or *Slave*. Radio communication will only work between *Masters* and *Slaves*. Only one unit on your network should be set to *Master*. It makes sense to make the base unit a *Master*, and all rovers *Slaves*. These settings will be handled automatically by the *Configure Base* and *Configure Rover* routines. So there generally isn't any reason to set the *Local Radio Type* manually.

The *Local Antenna Power Level* allows you to configure your radio to use more or less power. The less power the radio has, the less it will be able to communicate over longer distances. It may be useful to change the power level if you're rover is not travelling far from your base, and you're trying to conserve battery power.

Within the *Navcom Radio Setup* menu, you will be able to access the following status information for all visible Navcom units on the network:

External Power: Indicates whether the unit is plugged into an external power source(On or Off).

Battery A: Indicates whether a well charged battery is plugged into Battery Port A (On/Good or Off/Low)

Battery B: Indicates whether a well charged battery is plugged into Battery Port B (On/Good or Off/Low)

Status: Indicates whether the unit is sending out corrections. (BASE or ROVER)

If more than two units are present, you can access this information for the additional units by selecting the desired unit's radio ID from the *Remote Radio ID* dropdown menu.

Configure RTCM Submenu:

(Note: To access this menu, first configure the unit as a BASE and set the *Correction Type* to either *RTCM RTK* or *RTCM DGPS*.)

Choose *message 18/19* to make your RTCM RTK base broadcast RTCM message types 18/19.

Choose *message 20/21* to make your RTCM RTK base broadcast RTCM message types 20/21.

Choose message 1 to make your RTCM DGPS base broadcast RTCM message type 1.

Choose message 9 to make your RTCM DGPS base broadcast RTCM message type 9

Edit Base Position Submenu:

(Note: To access this menu, first configure the unit as a BASE)

If your BASE already has a GPS position set, it will be shown here. (If you don't see it, trying pressing Retrieve). To edit this value, change the displayed number and press the *Lock* button.

Click *Survey* to read a new GPS position from the Navcom unit.

Click *Empty*, to clear the GPS position from the unit.

Reset Unit Submenu:

Click *Soft Reset* to send a reset command to the Navcom unit. If the command is successful, all three status lights on the unit should go solid temporarily. After performing a soft reset, you will have to go to the *Configure Port* Submenu to reconfigure the control port.

Click *Factory Reset* to send an emergency reset command to the Navcom unit. However, in nearly all cases, it is only necessary to use the *Soft Reset* button. After performing a factory reset, you will have to go to the *Configure Port* Submenu to reconfigure the control port. *View Firmware* Submenu:

This submenu displays the Navcom firmware version your unit is using, along with the hardware serial numbers and the hardware model name.

Navigation Status Submenu:

If *Valid Navigation* reads *Yes*, your unit has successfully solved its position. If it reads *No*, the unit's position has not yet been calculated, and an error message explaining why will be displayed in the *Error* field. A rover will not try to use RTK corrections unless its navigation is valid. Similarly, a base will not broadcast correction unless its navigation is valid.

Navigation Status will read *AUTONOMOUS* if it is not receiving the type of corrections it has been configured to use. It will read FLOAT if it is receiving the right kind of corrections, but hasn't finished using them to calculate its position. It will read LOCK when it is receiving corrections and has successfully used them to calculate its position.

Navigation Mode displays the specific type of correction that is currently being used.

# of Satellites Used shows the number of satellites the unit is able to use in its solution.

All *DOP* values are also shown here (GDOP, PDOP, HDOP, VDOP, and TDOP).

Click *Refresh* to load the latest values from the device.

Monitor Incoming Corrections Submenu:

(Note: To access this menu, the local unit must be configured as a ROVER.)

This menu displays the number of seconds since the arrival of each RTK correction type. At the top, the correction type currently being used is displayed.

In *NCT Correction Mode*, the relevant messages are 5B (correction), which should be arriving every second, and 5C (base position), which should be arriving every 16 seconds.

In *CMR Correction Mode*, the relevant messages are cmr0 (correction), which should be arriving every second, and cmr1(base position), which should be arriving every 30 seconds.

In *RTCM RTK Correction Mode*, the relevant messages are RTCM message 22, and either messages 18 and 19, or messages 20 and 21, depending on your base's RTCM setup. Messages 18-21 should be arriving every second. Message 22 should be arriving every 6 seconds.

In *RTCM DGPS Correction Mode*, the age of correction messages (1 and 9) cannot be monitored here.

Click *Refresh* to load the latest values from the device. *Configure Base Submenu:*

Before clicking *Configure Base,* first choose the type of corrections you want to use. When you click *Configure Base*, all steps necessary to configuring a base will be performed. You will be prompted for a Base Position and a Radio ID. Upon completion, the unit status should read BASE. If it does not, or if an error occurs during base configuration, try again, or consult the Base/Rover configuration troubleshooting section below.

Configure Rover Submenu:

Before clicking *Configure Rover,* first choose the type of corrections you want to use. When you click *Configure Rover*, all steps necessary to configuring a base will be performed. Upon completion, the unit status should read ROVER. If it does not, or if an error occurs during rover configuration, try again, or consult the Base/Rover configuration troubleshooting section below.

Switching the device you're plugged in to:

Whenever you switch the device you're plugged into be sure to either close the GPS Setup window, or click *Retrieve Settings* from the top level GPS setup menu.

Troubleshooting Invalid Navigations:

If the *Navigation Status* menu reports an invalid navigation, you're unit has not yet been able to calculate it's position. The unit may need more time, if less than 4 satellites are visible, or an error is reported. If you can't get a valid solution for a few minutes, try raising the PDOP mask, or lowering the Satellite elevation mask.

Troubleshooting Base/Rover Configuration:

If you've configured a base to output corrections, and you're rover does not appear to be receiving the corrections, try each of the following in order:

1) Verify that your BASE and ROVER are both set to the same correction type.

2) Under *Configure Radio*, check that your ROVER is set to slave, and that your BASE is set to master.

3) Under *Configure Ports*, check that both your base and your rover's RTK Data Ports are set to the proper value (Usually *Radio Port*).

4) Under the *Edit Base Position*, check that your BASE is set to a valid position. Note that if the given position is too far away from the position the BASE is reading, the BASE will not send corrections.

5) If you're trying to use RTCM, make sure the BASE and ROVER have the same station ID's.

6) Try increase the RTK Max Age constraint.

7) Under *Navigation Status*, verify that the Navigation is valid on both units. If either unit does not have a valid position solution, correction wills not work.

8) Under *Monitor Corrections*, verify that the corrections you're using are arriving regularly. If they aren't you may need to reset both units.

9) Try configuring the base and rover again.

10) If all else fails, *Soft Reset* both units through the *Reset Unit* menu. After doing so, you will have to reconfigure the port settings of each device through the *Configure Ports* menu, and wait a few minutes for the devices to recalculate their position.

11) If none of these steps work, contact Carlson Software Technical Support.

Troubleshooting when you can't establish communication with the unit:

If all of your commands in the GPS setup menu are failing, try opening the *Configure Ports* submenu, selecting the proper Control Port, and saving the new settings. Make sure that you're plugged into the port you have chosen to be the control port.

If this does not work, issue a soft reset command. If this fails, try a factory reset command. If even this fails, call Carlson Software Technical Support.

Nikon Total Stations

Nikon A-Series includes the A5LG/A5, A10LG/A10 and A20LG/A20. Also the C-100 and D-50 have the same communication as the A-Series and should be used in the SET mode.

Nikon 500 Setup

1. Turn on Nikon

2. Turn it Horizontally and Vertically to set it.

3. Connect Nikon to AgStar

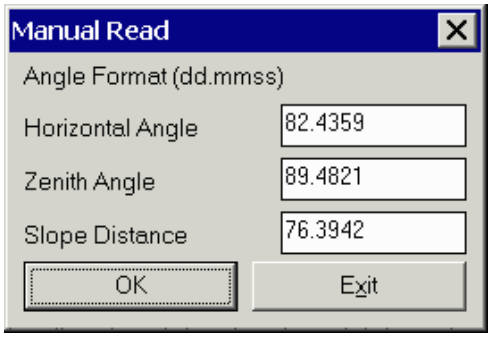
NOTE: 9-pin serial cable from Nikon to AgStar should be NGT type and not SOKTOP.

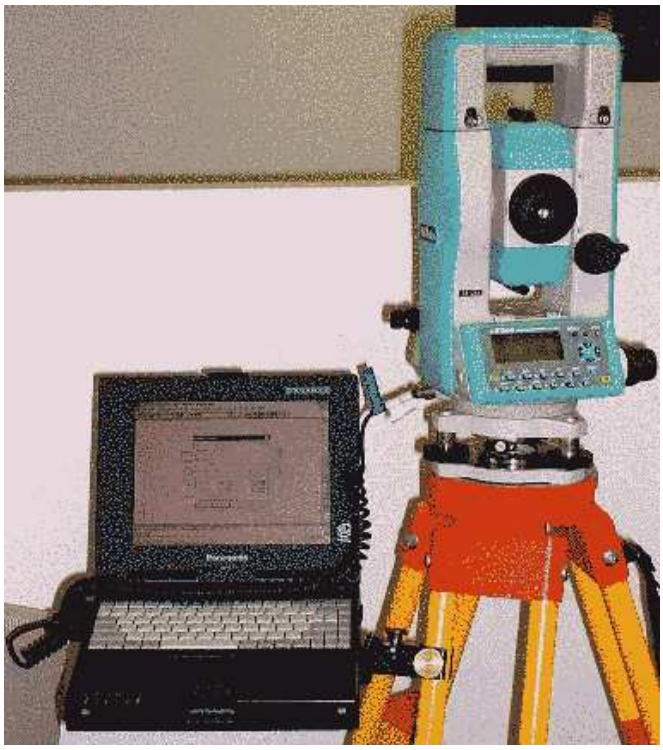
4. In AgStar go to Configure Survey, and under equipment type put Nikon 300,400,500 series.

5. To make sure the baud rate matches, under the Configure Survey menu click on General Settings and check the baud rate. On Nikon press [MENU], then 3 for "sett", and 6 for "comm". The baud rate can be changed using the arrow keys.

6. Exit the Configure Survey menu.

7. To check if units (Ft /M) matches for correct results, in AgStar under Inq-Set go to Drawing Setup and select the appropriate button. On Nikon, press [MENU] and 3 for "sett" again, but now press 5 for "unit".





Simulation GPS

Simulation GPS mode is for demonstration purposes to show or pratice AgStar functions. This mode allows you to run AgStar without being hooked up to any equipment. The program will automatically generate a position. This position is the first point in the alignment. If there is no alignment, then the starting point is 5000,5000,1000. There are keyboard commands to control the simulation position during continuous read commands such as Stakeout and Track Position. Here are the keyboard commands:

L - Turn Left

R - Turn Right

F - Go Faster

S - Slow Down

U - Up

D - Down

W - Switch Direction

Sokkia

Sokkia Radian IS

Hardware Setup 1) Make sure that the Radian IS has fully charged batteries installed, as described in the receiver documentation.

2) Connect the Radian IS serial cable to "COM1" on the Radian IS, plugging the other end into the controlling computer's serial port.

3) If the Radian IS is to be used as a base, connect a PDL base radio to the "COM2" port of the receiver. If the IS is to be used as a rover, connect a PDL rover radio to the "COM2" port if the receiver.

4) Power the Radian IS on with its external power switch.

5) Once the receiver finishes its self-initialization (when all the lights on the side panel go out and then the battery light lights in just one position), it is ready for use with AgStar. However, positions will not be able to be logged until the receiver has acquired a few satellites. The receiver has enough satellites when the center light is at the second or higher level (when it is orange instead of red).

Software Setup

6) To configure the IS for use, select "Equipment Setup" from the Survey pull-down menu. This will open a menu with several options:

a) Radio Baud Rate- This radio button sets the baud rate for COM2, the radio COM port. Make

sure this number and the number the PDL's are set for is the same.

b) Station Type-This sets whether the Radian IS is to be configured as a base station or a rover.

c) Elevation Type-This allows selection of Geoid (MSL) or Ellipsoidal measures for height/altitude.

d) RTK Dynamics- This sets the dynamics mode of the receiver. In general, this setting should be set to "Dynamic/Kinematic"

e) Message Type-sets what format of corrections this receiver will send/receive for RTK.

f) Motion Dynamics-Used to set the receiver's calculations appropriate to the motion of the receiver. g) Elevation Mask-This is the satellite elevation cutoff. No satellites with elevation less than this number will be used in corrections. This allows filtering out of satellites close to the horizon, which provide less accurate calculations for positions.

h) Send Command to Receiver- Allows a specific user-entered command to be sent to the receiver. Mostly used for troubleshooting with Technical support.

i) Configure Base- Configures the parameters of a base station for the receiver (Ex: Current position, etc.)

j) Power Cycle Receiver- powers the receiver down and then turns it back on, clearing the main memory.

k) Save and Exit-save all settings changes and exit this menu.

l) Cancel-Restore original settings and exit this menu.

To set the Radian IS up as a Rover:

7) Select "Rover" for Station Type, and set the Radio Baud to match the PDL's which are being used. Also, set "RTK Dynamics" to "Dynamic/Kinematic", and set Motion Dynamics to the appropriate option.

8) Select "Exit and Save". The receiver is now ready for use as a rover.

To set the Radian IS up as a Base:

7) Select "Base" for station type, and set the Radio Baud to math the PDL's which are being used.

For most jobs, set RTK Dynamics to "Dynamic/Kinematic" (unless you are *sure* that static is more appropriate-even small fluctuations from wind on the pole can cause problems in Static mode). Set motion dynamics to Foot/Walking, and then select "Configure Base Station"

9) In the menu dialog that opens, there are a few buttons:

a) Read from GPS-Read a position from the GPS and fix to that position

b) Enter Lat/Lon-Fix to a manually entered Lat/Lon position

c) Enter State Plane Coord- Fix to a manually entered State Plane Northing/Easting position

d) Read From File- Fix to a position read from a $*$.ref file.

e) Cancel-Cancel base setup

If Read From GPS is selected, the software will read once from the GPS receiver, and then fix to that position. If Enter Lat/Lon is selected, a dialog box will open and a Latitude and Longitude must be input manually. If Enter State Plane Coord is selected, a dialog box will open allowing the input of a set of Northing/Easting coordinates by hand. Read from File will open a File->Open dialog and ask for the file name of the file to open.

Regardless of which option is selected, after the position is determined, this position will be displayed, and dialog boxes will open to enter a station id and the measured base antenna height. Once these values are entered, base setup is complete and the "Exit and Save" button can be selected to exit the GPS Setup menu.

Sokkia 500 Series

1) Turn on Sokkia

2) Turn it Horizontally and Vertically to set it.

3) Connect Sokkia to AgStar

4) In AgStar go to Configure Survey, and under equipment type put Sokkia

5) To make sure the baud rate matches, under the Configure Survey menu click on General Settings and check the baud rate. On Sokkia press [ESC], then [CNFG]. Scroll down or enter 4 for "Comms setup." The baud rate can be changed using the arrow keys, when done press [ESC].

6) Exit the Configure Survey menu.

7) To check if units (Ft /M) matches for correct results, in AgStar under Inq-Set go to Drawing Setup and select the appropriate button. On Sokkia, in [CNFG] scroll to or enter 5 for "unit" and select appropriate unit using the arrow keys.

Topcon Total Stations

The Topcon instrument must have CR/LF (carriage return/linefeed) turned on for communication with AgStar.

To set this with 200 series:

1. Turn instrument off
2. Turn instrument on while holding F2 key
3. Choose F3 (Others set)
4. Press F4 (Page down)
5. Choose F3 (CR/LF) and set it on

To set this with 700 series:

1. Choose Parameter from the main screen
2. Scroll down until you find CR/LF and set it on

Topcon GTS-700

To set the instrument to work with AgStar, press [F2] for "std" on the instrument.

Topcon 800-A Remote Setup

Topcon Setup:

1. Turn on the Topcon

2. Connect the Topcon to one of the radios, and the other radio connect to AgStar

3. Under the Survey menu go to Configure Survey, and under equipment type select Topcon800A-Remote.

4. To set Topcon for external mode Press [F1] for "prog", then [F6] for "more". This will lead to more programs. Enter [F2] for "Ext.Link."

5. To select the radio channel, in External Link enter 2 for "settings" and 4 for "parameter (radio modem)", then 3 for "set channel". Using the arrow keys change the channel. When done press for [F1] for set, then press [ESC] until get back to External Link Menu.

NOTE: Channel on the Topcon should match the channel set in AgStar.

6. After channel is set press 1 for "Execute"

7. Topcon is ready.

NOTE: If the batteries are low either in Topcon or the radios, communication problems will arise.

AgStar Setup:

1. In Configure Survey, under equipment type there should be Topcon800A-remote. In General Settings Baud Rate should be set to 9600.

2. After Configure Survey go to Total Station Setup and make sure the radio channel or radio frequency matches the channel and frequency in AgStar. Press Ok when done.

Topcon 800A Quick Lock

1. Dismount the handle from the Topcon, and mount RC-2H. Secure it with the fixing screw.

2. Attach RC-2R to the prism, and turn it on.

3. Using the Y cable attach the RC-2H to the radio and AgStar.

4. In Joystick click on Quick Lock and Topcon will do angle turn until it finds a prism in which it will lock to, and will start tracking.

5. If RC-2H is not attached to the radio with Y cable, when Quick Lock is pressed the big yellow button on RC-2H needs to be pressed in order for the Topcon to search for the prism.

Trimble

Trimble NT300D 1) In order to properly configure the NT300D to work with AgStar, it must first be powered up in Setup mode (by holding down the [Setup] button on the front panel of the receiver while powering it on) so that the advanced setup options are available. Once the NT300D is powered up in this mode, bring up the Setup menu via the [Setup] button. Page down using the *More* menu option until the *I/O* menu item is available, and select it.

a. In the I/O menu, select whichever port is to be used to interface the receiver with the computer running AgStar (Port 1 by default). Next, set both the input and the output to transmit/receive in *NMEA*, at *9600* baud rate. The final option, Remote Select, should be set to *Primary*.

b. Now the NMEA sentences must be configured. From the I/O menu, enter the *NMEA Sentences* submenu. Disable all sentences, save for the *GGA* sentences and the *GSA* sentences. Ensure the Talker ID is *GP*. From here, *Return* to the I/O menu.

c. The *NMEA Control* menu item, reachable from the I/O menu, has three options. The Output Rate here should be set to *1 second*, the Position Output Rate set to *Output Rate*, and the NMEA Output Version to *2.1*.

Next, the GPS settings must be configured, and can be found in the *GPS* menu under the main *Setup* menu.

The GPS Mode should be set to *3D*, and the DGPS mode set to *Auto*. The DGPS source should be toggled to *Internal*, and the Pos/Vel Filter should be *Off*. Mask Values should be left at *Default*, and the SNR at *M*.

Finally, the Beacon Receiver configuration (under *Beacon Receiver* on the Setup Menu) needs to have its Search Mode set to *Auto-Dist* Mode. All other values in all menus ought to either be left at their default settings, or configured as necessary to the local conditions (in the case of antenna height, etc.).

2) The RMS value reported in AgStar is the RMS value of the standard deviation of the range inputs to the navigation process including pseudoranges and DGPS corrections.

The NT300D is now properly configured, and if connected to a computer running AgStar, will transmit position fix data to the computer automatically. Before using it, however, it is best to power it down and then turn it back on normally, as running it in Setup Mode is not recommended.

Trimble 4000 Series

Hardware Setup:

1. Setup the antenna and GPS receiver as normal. The radio should be on I/O Port 2.

2. Connect the Computer that AgStar is running on to I/O Port 1 by the appropriate cable.

Front-Panel Configuration:

Base Station:

1. After powering on the receiver, press the [Control] Button. From the selections available, select **MORE**. This will bring up a second page of options. Select **MORE** again. The front panel screen should now be on **RECEIVER CONTROL** "3 of 7".

2. Select **BAUD RATE/FORMAT**, and from the menu that this creates, select **SERIAL PORT 1 SETTINGS**.

3. Ensure that the port is set to 38400 baud, 8-Odd-1 Format, with no flow control.

4. Similarly, make sure that the settings for I/O Port 2 agree with those of the type of radio being used (typically 9600 8-None-1).

5. Return to the **RECEIVER CONTROL** menu, and go to page 4 o 7. Select **REFERENCE POSITION**.

6. Enter the Lat/Lon of the position the base is located at. Alternately, select **HERE** to have the GPS unit read the current position and use that as the base reference point.

7. On page 1 of the **RECEIVER CONTROL** menu, select **RTK OUTPUT CONTROL**.

8. Set the **RTK OUTPUTS** to Port 2, and the **ANTENNA HEIGHT** to the measured height of the antenna.

9. Ensure that all other forms of output (Cycled Output, 1PPS output, Event Markers, etc.) are disable. These options may all be accessed with the submenus accessible through the [Control] button.

10. Ensure that the Synch time of the Rover and Base are the same. This setting may be accessed by first pressing [Control] and then cycling through the menus until the **MASKS/SYNCH TIME** option is available

Rover Station:

1. After powering on the receiver, press the [Control] Button. From the selections available, select **MORE**. This will bring up a second page of options. Select **MORE** again. The front panel screen should now be on **RECEIVER CONTROL** "3 of 7".

2. Select **BAUD RATE/FORMAT**, and from the menu that this creates, select **SERIAL PORT 1 SETTINGS**.

3. Ensure that the port is set to 38400 baud, 8-Odd-1 Format, with no flow control.

4. Similarly, make sure that the settings for I/O Port 2 agree with those of the type of radio being used (typically 9600 8-None-1). 5. Return to the **RECEIVER CONTROL** menus, and go to page 2.

6. Select **RTK ROVER CONTROL**.

7. Toggle the **ENABLE** setting to **L1/L2**.

8. Push the [Status] button, and select **POSITION**. There should now be an **RTK** option. Select it. This will bring up a screen displaying delta Northing/Easting, correction status, etc.

9. Ensure that the **STATIC** option appears at the right. This means you are in kinematic/rover mode. If instead the **ROVE** option is available, select it.

10. Ensure that all other forms of output (Cycled Output, 1PPS output, Event Markers, etc.) are disable. These options may all be accessed with the submenus accessible through the [Control] button.

11. Ensure that the Synch time of the Rover and Base are the same. This setting may be accessed by first pressing [Control] and then cycling through the menus until the **MASKS/SYNCH TIME** option is available.

Trimble 4700/4800

Hardware and Equipment:

1. Make sure that the computer's serial port is connected to the 4700/4800 in it's COM1 port (typically the port that a data collector is normally plugged into). Power should be supplied on COM2, and any radio used for RTK should be plugged into COM3.

2. All other equipment (antenna, wires, etc.) should be set up as normally directed by the manuals.

Software Configuration:

1. After selecting the Trimble 4700 equipment type from the "configure survey" menu, open up "Equipment Setup." This should bring up a new window/dialog box with the following options:

a. Receiver Type- Select whether you are using a 4700 or 4800 receiver.

b. Station Type- Choose what type of RTK station you are setting this receiver up as- a base or rover.

c. RTK Correction type- Select the type of Corrections you would like a base station to transmit. Note that CMR messages should be used for most precision applications, as RTCM is only capable of producing less-accurate floating precision positions

d. Radio Baud Rate- The baud rate of the radio port. This should be left at the default setting of 9600 in general

e. Satellite Elevation Cutoff- All satellites with elevation from the horizon of less than this numeber will *not* be used in calculating a position. This allows less accurate low elevation satellite to be factored out of a position.

f. Configure Base Station- Will configure the receiver to act as a base. See "Configuring the Base Station" below.

g. Cancel without saving- will exit this menu without saving any changes that have been made.

h. Save and Exit- Will save these settings to the reciever and to AgStar's setup and exit out of this menu.

Configuring Rover: No real configuration is necessary, aside from setting up the equipment and setting the appropriate Receiver Type, Station Type, and Satellite Elevation Cutoff.
Configuring Base Station: 1. After selecting all the appropriate settings in "Configure GPS," click on the "Configure Base Station" button.

2. In the menu dialog that opens, there are a few buttons:

a. Read from GPS-Read a position from the GPS and fix to that position

b. Enter Lat/Lon-Fix to a manually entered Lat/Lon position

c. Enter State Plane Coord- Fix to a manually entered State Plane Northing/Easting position

d. Read From File- Fix to a position read from a $*$.ref file.

e. Cancel-Cancel base setup

If Read From GPS is selected, the software will read once from the GPS receiver, and then fix to that position. If Enter Lat/Lon is selected, a dialog box will open and a Latitude and Longitude must be input manually. If Enter State Plane Coord is selected, a dialog box will open allowing the input of a set of Northing/Easting coordinates by hand. Read from File will open a File->Open dialog and ask for a file name of a reference file ($*$.REF) to open for use in corrections.

Regardless of which option is selected, after the position is determined, this position will be displayed, and dialog boxes will open to enter a station id(used by the base to identify iteself to the rover(s)) and the measured base antenna height. Once these values are entered, base setup is complete and the "Exit and Save" button can be selected to exit the GPS Setup menu. At this point, whenever looking at a menu that displays the connection status, "REFERENCE" will be displayed, instead of Float, Fixed, or Autonomous.

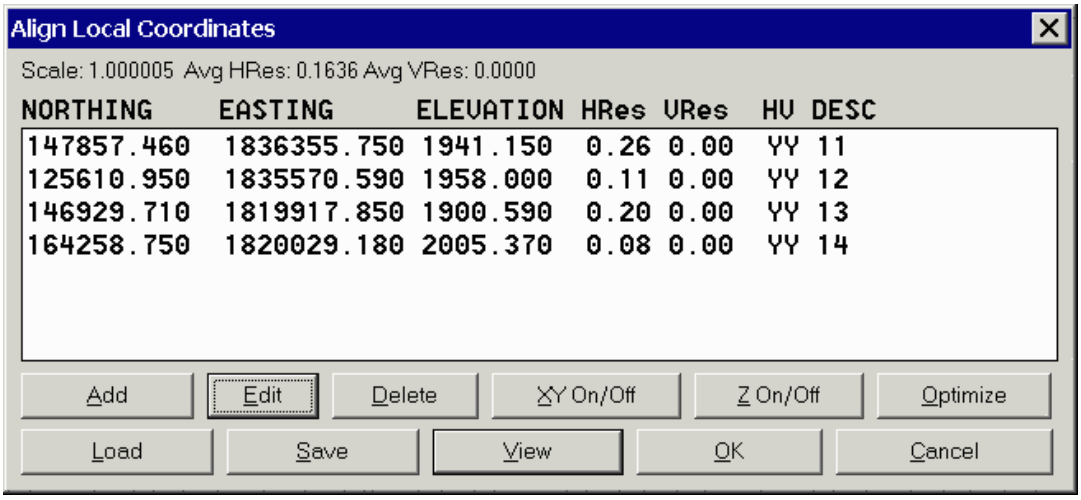
Align GPS To Local Coordinates

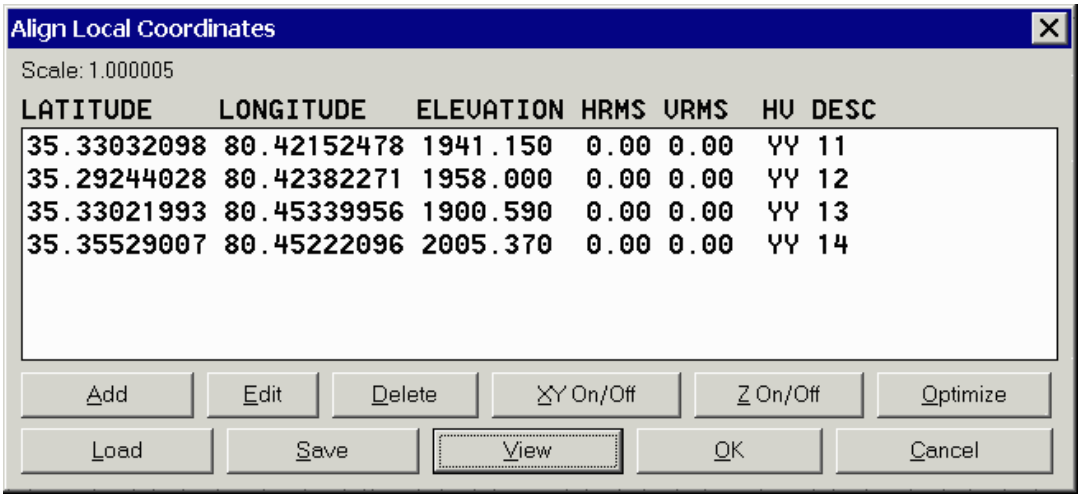
AgStar reads a latitude, longitude and height position from the GPS rover receiver and converts these values to State Plane or UTM coordinates for the current zone as set in Configure Survey. Using local coordinates and their corresponding GPS position, Align Local Coordinates applies a transformation to convert the state plane or UTM coordinate to the local. AgStar can operate in three different modes depending on the Align Local Coordinate settings:

1) No points - No Adjustment

2) One point - Translation Only

3) Two or more points - Translate, Rotate and Scale





Without any alignment points set, AgStar will operate with no alignment which directly uses the state plane or UTM coordinates. In order for the coordinates to be the true state plane coordinates in this alignment mode, the GPS base receiver must be set up over a known point and the true Lat/Long for the point must be entered in the base as the base position. Otherwise, if the base is set over an arbitrary point, then the coordinates will not be true state plane.

In one point alignment mode, one pair of GPS and local coordinates is specified. The differences between the GPS and local northing, easting and elevation for these points are used as the translation distances in the transformation. The rotation will use either the state plane grid or the geodetic as north. No scale is applied in this transformation.

A two or more point alignment is used to align to an existing local coordinate system. At least two pairs of local and GPS coordinates must be entered.

In addition to the northing and easting transformation, SurvStar will also translate the elevation from the GPS system to the local. The elevation difference between the two systems is modeled by a best-fit plane.

An alignment is only valid if the base receiver setup has not changed since the alignment points were recorded. In order to use an alignment when returning to a site, you must set up the base receiver in the same position and enter the same LAT/LONG coordinates for the base.

The *Align GPS to Local Crds* menu item brings up the alignment dialog box. There is more information than to fit in one window, so use the *View* button to switch between viewing the local coordinates and the GPS Lat/Lon.

Each line in the box represents one alignment point. Each point in an alignment file relates a specific Lat/Lon/Elv to a specific Northing/Easting/Elevation for your local coordinate system. AgStar will use the current alignment file every time that the GPS is read. It provides the necessary adjustment to properly convert that position to your coordinate system.

In the local points view, the HRes column shows the horizontal residual and the VRes column shows the vertical residual. The residual is the difference between the actual point and the point calculated using the alignment transformation. In GPS points view, the HRMS and VRMS columns show the horizontal and vertical RMS values when that point was recorded.

The *On/Off* buttons allow you to switch whether the highlighted point is used for the horizontal and/or vertical alignment. The *HV* column shows a ‘Y' if this point is used in the calculations. Otherwise it shows an ‘N'. The H column represents horizontal control and the V column vertical control. For example, you may wish to use 2 points for horizontal alignment and one for vertical.

The *Optimize* button will find the combination of turning alignment points on/off for horizontal and vertical such that the horizontal and vertical residuals are minimized.
The *Desc* field shows an optional description of the alignment points.

The scale factor and average horizontal and vertical residuals appear at the top of the window. These values serve as a check that the alignment is valid. The scale factor factor should be closed to 1.0 (in range of 0.9 to 1.1). The average residuals should be less than 0.2.

XY On/Off toggles the highlighted alignment point horizontal component off or on. Alignment points with the horizontal component toggled off will not use the northing and easting of that point for adjustment calculations.

Z On/Off toggles the highlighted alignment point vertical component off or on. Alignment points with the vertical component toggled off will not use the elevation of that point for adjustment calculations.

Note: When you toggle either the XY or Z component off or on for any alignment point the scale factor and Horiz/Vert residuals are recalculated automatically. Breifly toggling XY or Z components off or on and reviewing the scale factor and residuals changes is a quick approach to finding the best alignment points. AgStar can handle an unlimited number of alignment points.

Highlight an existing alignment point entry and pick *Delete* to delete that alignment point.

Pick the *Add* button to create an alignment point. The Add Alignment Point dialog box appears. There are two ways to enter the local coordinate points: by entering the N, E, Z, or by using an existing point number stored in the current coordinate CRD file. The GPS values can also be specified by two methods: by entering in the Latitude, Longitude and Height or by occupying the control point with the rover and taking a GPS reading at this location. Manually entering the Lat/Lon can only be done when the base is setup on a known location using a true lat/lon position. Otherwise AgStar needs to use the Read GPS method. For this method, the base can be setup with a lat/lon that only needs to be close (within 100 feet) of the actual lat/lon. This type of position can be read from an autonomous GPS posiiton. With the base setup on this approximate lat/lon, go with the rover to the control points and use the Read GPS option in the Add function. The rover GPS solution must be in "fixed" status when the alignment point

is added. By reading the rover GPS position for the alignment points, the alignment will transform the coordinates from the GPS system of the current base setup to your local coordinate system.

Load allows you to open an existing alignment file. Only one alignment file can be open at a time. Alignment files have a DAT extension and stored in the Data directory by default.

Save stores alignment files (DAT extensions) to a file. Files are by default stored to the Data subdirectory.

The *OK* button will set the current alignment to the settings in the dialog.

Alignment Methods

AgStar can operate by the following Alignment methods:

Alignment Method 1) - No alignment points

Alignment Method 2) - One point alignment

Alignment Method 3) - Two or more alignment points

Alignment Method 1

With no alignment of the rover, AgStar will report Northing and Easting as State Plane or UTM coordinates. In order for this method to give accurate State Plane or UTM coordinate values, the GPS base receiver must be set up over a known point and configured using the true Lat/Long/Hgt or true State Plane coordinates. If the base is set over an arbitrary point, configured by reading the GPS, the RTK GPS stored coordinates will be translated up to a 200 feet but accurate in relation to each other.

When using this method, you can skip *Align GPS to Local Crds* and start surveying immediately once the base is configured and transmitting its position and the rover is fixed.

In most cases, you cannot use Method 1 because you will not have setup the base on a point whose exact true position you know. Therefore the base corrections are going to be off a certain distance north/south and a certain distance east/west. This is why you want to do an alignment. You are showing AgStar how to correct for the north/south and east/west offsets. Any points surveyed with the alignment file active will be translated to their proper position.

To gather alignment points, you put the GPS antenna over a point with known coordinates and AgStar records the GPS Lat/Lon/Elv and the Northing/Easting/Elevation you give it. This point can be a local coordinate, for example a stake you are calling 5000,5000. It can also be a true State Plane point. Using one or more State Plane points will give you an alignment to true State Plane (even if your base is not using its own true position.)

Alignment Mehtod 2

This method uses one alignment point to translate the GPS coordinates to local or true State Plane coordinates.

Remember that if the base is set up over an arbitrary point, the GPS coordinates can be off from true state plane by up to 200 feet. This alignment method can be used to correct for this by translating the system onto true state plane coordinates.

You can choose if you want the coordinate system North to be Geodetic North or State Plane Grid North under Configure Survey>GPS Settings. If you specify a scale factor in that dialog box, it will be applied to all points recorded.

One point alignment is useful for data collection on a new site. In this case you can set up the GPS base receiver anywhere convenient. Then position the rover over a point (preferably one you can find again) and add this point as your one alignment point by reading the GPS point and

entering a local coordinate like 5000,5000,100. Now the local coordinate system is set around this first point at 5000,5000,100.

This method is commonly used for small topo or stockpile RTK GPS surveys. When collecting or staking data at distances greater than 2 miles from the base, both the horizontal and vertical errors will begin to increase gradually. Therefore, you should use a multiple point alignment for large projects.

Alignment Method 3

This method is useful if you are arriving on a job which has already been surveyed. It assures that your survey is in the same coordinate system as the original survey.

Using control points, this method transforms the GPS coordinates to an existing local coordinate system. This method takes pairs of GPS coordinates and the corresponding local coordinates to define the translation, rotation and scale of the alignment.

In Configure Survey>GPS Settings, there is a choice for the transformation as Plane Similarity or Rigid Body. Plane Similarity will apply a scale factor to the transformation. The scale factor will be based on the alignment points and should always be very near 1.0 to be correct. The Rigid Body option will align by translate and rotate but no scale. Any difference in scale between the GPS and local coordinate systems will be distributed equally between the two alignment points. These differences will appear as horizontal residuals in the Alignment dialog.

Two pairs of points are sufficient to define the translation, rotation and scale for the transformation. But adding more alignment points yields the most accurate results for aligning to existing coordinate systems. Since two pairs of coordinates are sufficient to define the transformation, there is extra data when there are three or more pairs. The program uses a least-squares best-fit routine to find the transformation that minimizes the residuals. This one best-fit transformation is used to convert from the GPS to the local coordinate system for all the points. The residuals are the differences between the transformed GPS coordinates and the actual local coordinates.

A multiple point alignment is especially helpful on a survey which covers a large area. The error in raw GPS coordinates increases as you get farther from the base. Taking alignment points around the perimeter of your job site as alignment points will give you the best geometry for the alignment.

Align to Benchmark

If the base is moved for any reason, or if significant time has passed since the Master Benchmark was surveyed, it may be necessary to align to (or "tie-in to") a benchmark. Be sure your vehicle is positioned in exactly the same place in the field it was when you initially surveyed the benchmark.

Steps

1 You will be prompted to select a benchmark from the drawing. Click on the benchmark you want to align to.

2

With your vehicle positioned over the benchmark point, press the *Read* button. When the readings complete, click *Ok* on the *Read Average GPS* window.

3

You will be asked to select a number of GPS readings to be taken (10 is enough). Click *Ok* to begin readings. The coordinate system is now aligned to the chosen benchmark.

Pull-Down Menu Location: Survey
Prerequisite: A master benchmark, and preferably, fixed RTK
Keyboard Command: align_bmark

Typical Alignment Scenarios

Scenario: New site. In this case, there are no established coordinates on the site.

Alignment: Choose a point on site and do a one point alignment. For the local alignment point, enter the coordinates that you would like to use (ie 5000,5000,100). Under Configure AgStar->GPS Settings, The One Pt Align Azimuth option chooses between using true north (geodetic) or state plane north (grid). To use real world ground distances, set the Project Scale

Factor under Configure Survey->GPS Settings. Otherwise the default scale factor of 1.0 will collect points on state plane distances.

Scenario: One known state plane coordinate and you want to work in the state plane coordinate system.

Alignment: Either setup the base over the known state plane coordinate or do a one point alignment on this known state plane point. In Configure Survey->GPS Settings, set the One Point Align Azimuth to Grid and set the scale factor to 1.0.

Scenario: Multiple known control points.

Alignment: Choose two or more control points to align to. It is best to use control points around the perimeter of the site. Use as many control points as are available or enough to envelope the site. In Configure Survey->GPS Settings, set the Transformation to Plane Similarity to fit the GPS points onto the control points and set the Project Scale Factor to 1.0. After making the alignment, stake out another control point (ideally one the is not used in the alignment) to make sure the alignment is good.

Survey Benchmark

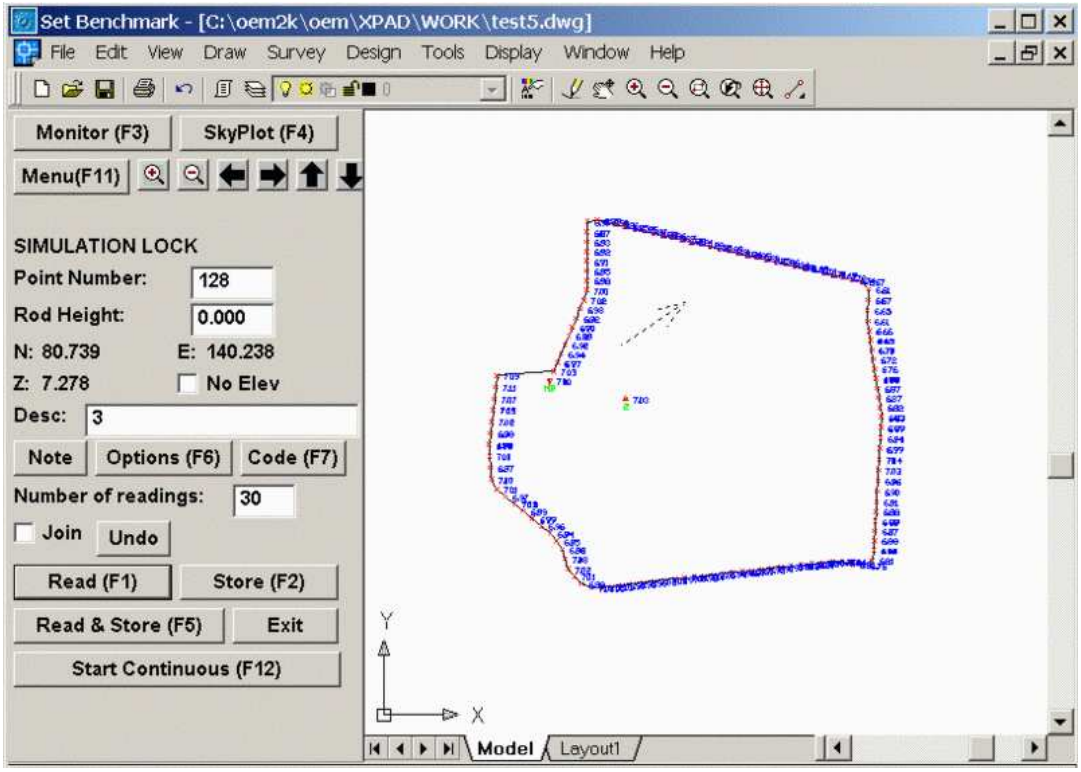
It is common practice to put intersperse five or more benchmark points across your field. Benchmark point surveys are more accurate then interior or perimeter surveys, because they are taken while motionless, and because they average over many GPS readings. After surveying a benchmark, you may want to mark it with a flag, in case you return to the field later, and need to realign to a benchmark (see *Align to Benchmark*).

Steps

1 Set the *Number of Readings* field to the number of readings to be taken and averaged (10 is enough).

2

With your vehicle positioned over the benchmark point, press the *Read* button. When the readings complete, click *Ok* on the *Read Average GPS* window.



3 Press the *Store* button, then press *Exit*. You'll probably want to mark this point with a flag.

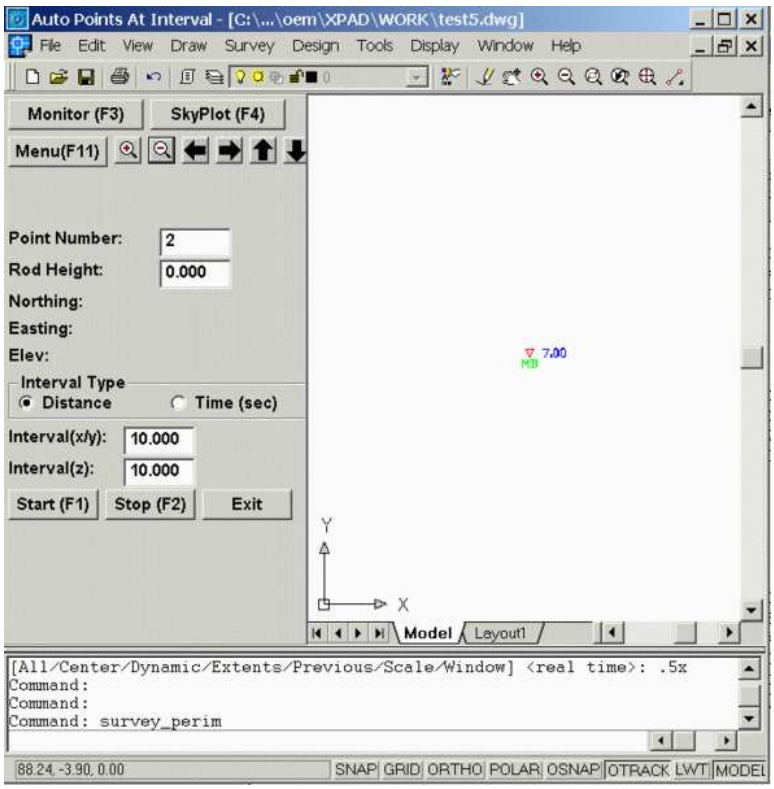
Pull-Down Menu Location: Survey
Prerequisite: A master benchmark, and fixed RTK
Keyboard Command: store_pt

Survey Perimeter

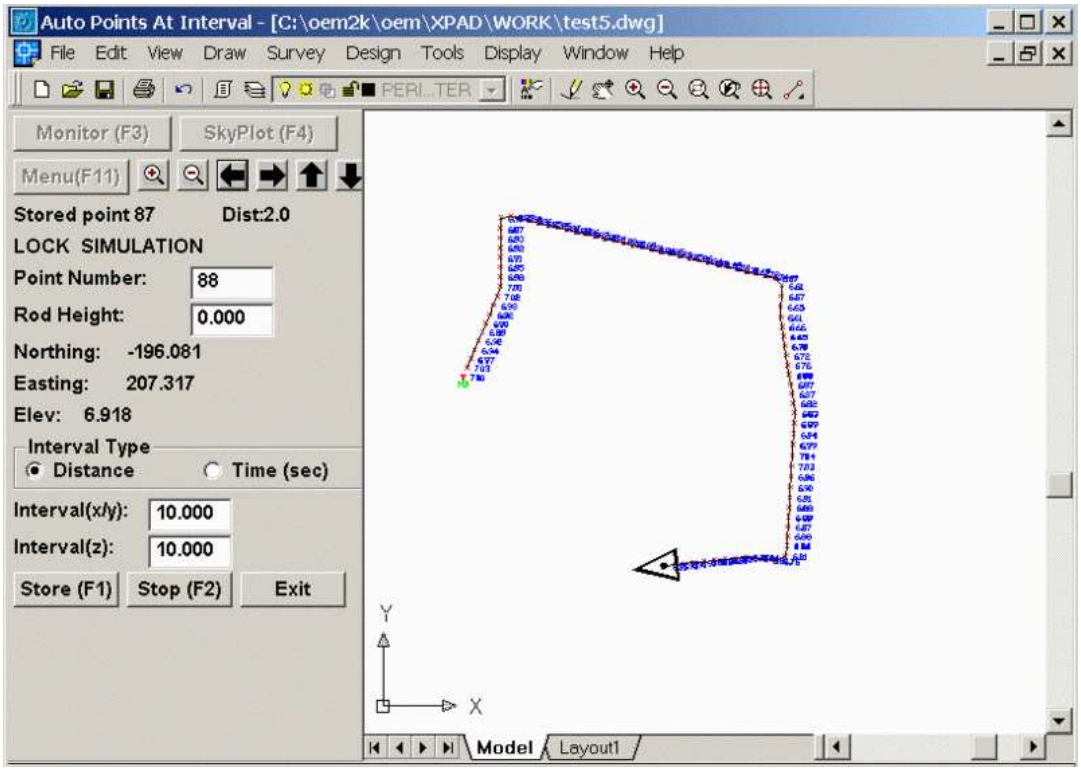
Before surveying the interior of the field, you must first survey the perimeter (border). Any surveyed points outside of the perimeter will not be factored into the field design.

Steps

1 To survey a point every time a specified horizontal distance is travel, select *Distance* as the interval type, and type the desired interval in the *Interval(x/y)* field. Alternatively, you can specify an amount of time between surveyed points by selecting the *Time* interval type.



2 Now, with your vehicle positioned at the edge of the field, press *Start* and begin piloting your vehicle around the field.



3 If you need to stop in the middle, press *Stop*, and when prompted to close the perimeter, press *No*.

4 When the survey is complete, press *Stop*, and choose *Yes* to close the perimeter. Then click *Exit*.

Pull-Down Menu Location: Survey
Prerequisite: A master benchmark, and preferably, fixed RTK
Keyboard Command: survey_perim

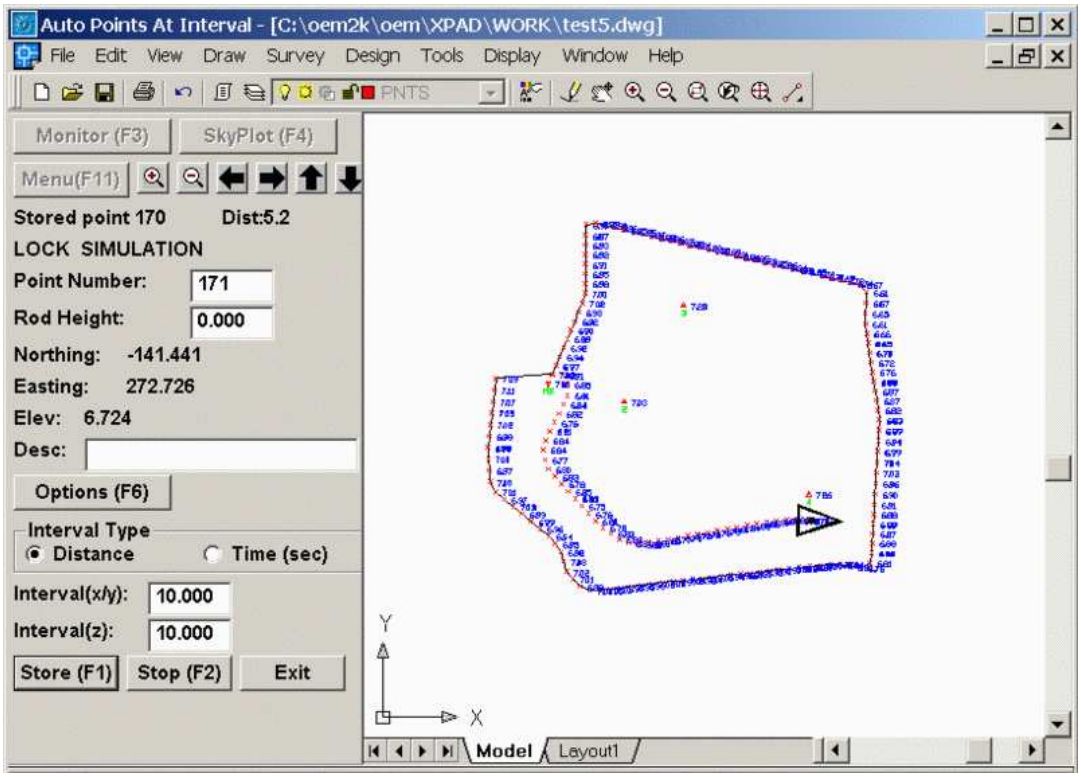
Survey Interior Surface

To get an accurate model of the field, a larger number of points within the interior of your field must be surveyed. Normally, it is sufficient to have ten to twenty feet separating your survey points; however, if there is a lot of variation in the slope of the field, you may need the points to be closer together.

Steps

1 To survey a point every time a specified horizontal distance is travel, select *Distance* as the interval type, and type the desired interval in the *Interval(x/y)* cell. Alternatively, you can specify an amount of time between surveyed points by selecting the *Time* interval type.

2 Press *Start* and begin piloting your vehicle back and forth across the field.



Try not to leave any unsurveyed patches, especially in areas where the terrain's natural slope changes.

3 If you need to stop in the middle, press *Stop*. If you need to add a benchmark, press *Stop,* then press *Add Benchmark*, and follow the instructions in the *Survey Benchmark* section of this manual. When complete, click *Exit*.

Pull-Down Menu Location: Survey
Prerequisite: A closed perimeter, and fixed RTK
Keyboard Command: auto_pts

Survey Subdivision line

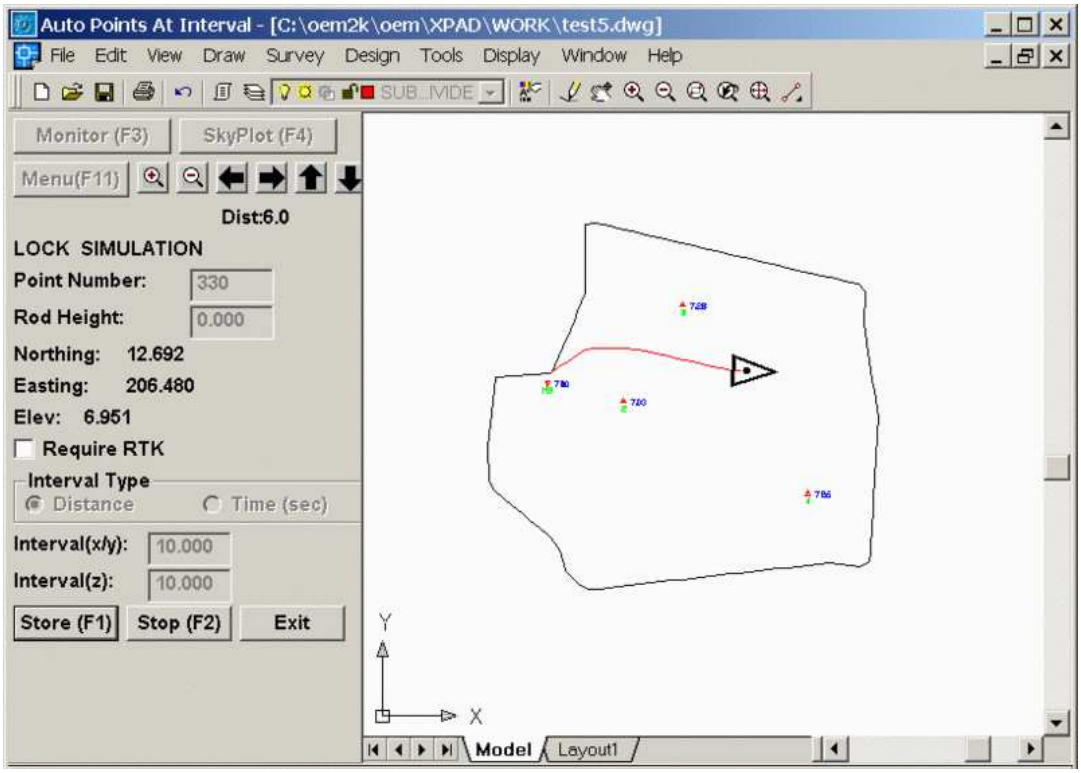
You may wish to divide your field into multiple sub-fields, in order to assign different designs to different areas. There are two ways to divide up your field: To draw the subdivision line with your mouse, see instructions on the *Draw Subdivision Line* feature. To draw the subdivision line with your vehicle, using GPS, select *Survey Subdivision Line* from the *Survey* menu.

When surveying a subdivision line, no new points are added to the drawing, so performing this function without RTK will not compromise the accuracy of the data.

Steps

1 If you do not have an RTK base set up, uncheck the *Require RTK* box.

2 Press start, and begin to pilot your vehicle across the field.



3 When you've reached the other side of the field, press *Stop*. Then press *Exit*. Note that after pressing stop, the subdivision is complete, and that you can't continue it later, as you can with during perimeter and interior surveys.

4 You will now be prompted to assign names to the two new areas. See instructions on *Assign Subdivision Area Name* for what to do next. Note that you must assign a name to any new section before you can design it.

Pull-Down Menu Location: Survey
Prerequisite: A closed perimeter, and preferably, fixed RTK
Keyboard Command: survey_subdivide

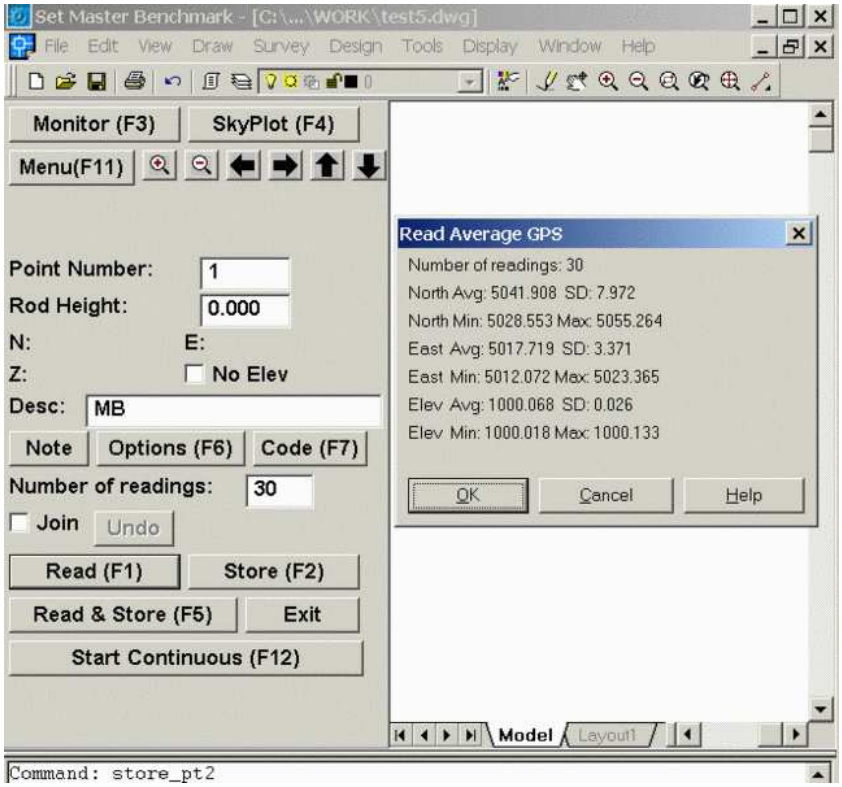
Survey Master Benchmark

A master benchmark point must be surveyed before any other points. Its purpose is to define the origin of your local coordinate system. When you survey a master benchmark, a mapping from GPS coordinates to your local coordinate system is created. For more complex coordinate mapping functionality, see the *Align to Local Coordinates* command.

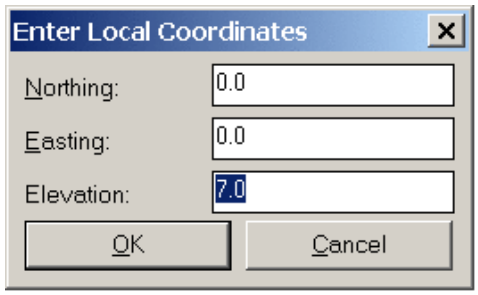
Steps

1 Set the *Number of Readings* field to the number of readings to be taken and averaged (10 is enough).

2 With your vehicle positioned over the benchmark point, press the *Read* button.



3 When the readings complete, click *Ok* on the *Read Average GPS* window. Press the *Store* button. You will be prompted for the desired coordinates of your master benchmark.



4 Enter the coordinates, click *Ok*, and then *Exit.* You may also want to mark the benchmark with a flag, in case you return to the field later, and need to realign to a benchmark (see *Align to Benchmark*).

Pull-Down Menu Location: Survey
Prerequisite: A drawing name, and fixed RTK
Keyboard Command: store_pt2

Monitor GPS Position

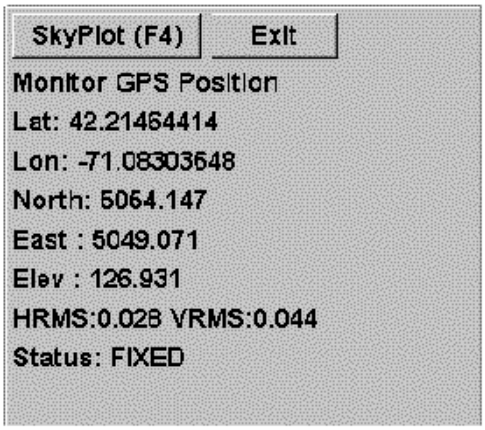
This command reports the current GPS Lat/Lon, local coordinates and GPS solution status. The latitude and longitude are reported in the DD.MMSSSSSS format. In this example, the latitude is 42 degrees, 21minutes, 46.4414 seconds north. The longitude is 71 degrees, 8 minutes and 31.5699 seconds west. Negative longitudes indicate longitudes west.

The next three items are state plane or local coordinates depending on the transformation in the Align Local Coordinates command. The *HRMS* and *VRMS* are measures of the reliability of the position that the receiver has calculated. They correspond to the position horizontally and vertically, respectively. If the receiver is autonomous, not receiving

corrections from a base, the RMS can be up to a few hundred feet. If this rover is computing a "Fixed" position, the RMS values should be less than one foot, probably close to a tenth of a foot. If the receiver looses the fix and becomes "Float", the RMS values will jump to between one and ten feet.

Depending on the type of GPS receiver, the Monitor screen will also show more values like radio link status and receiver battery status.

The *Skyplot* button will jump you to that window so you can see the satellites the receiver is using.

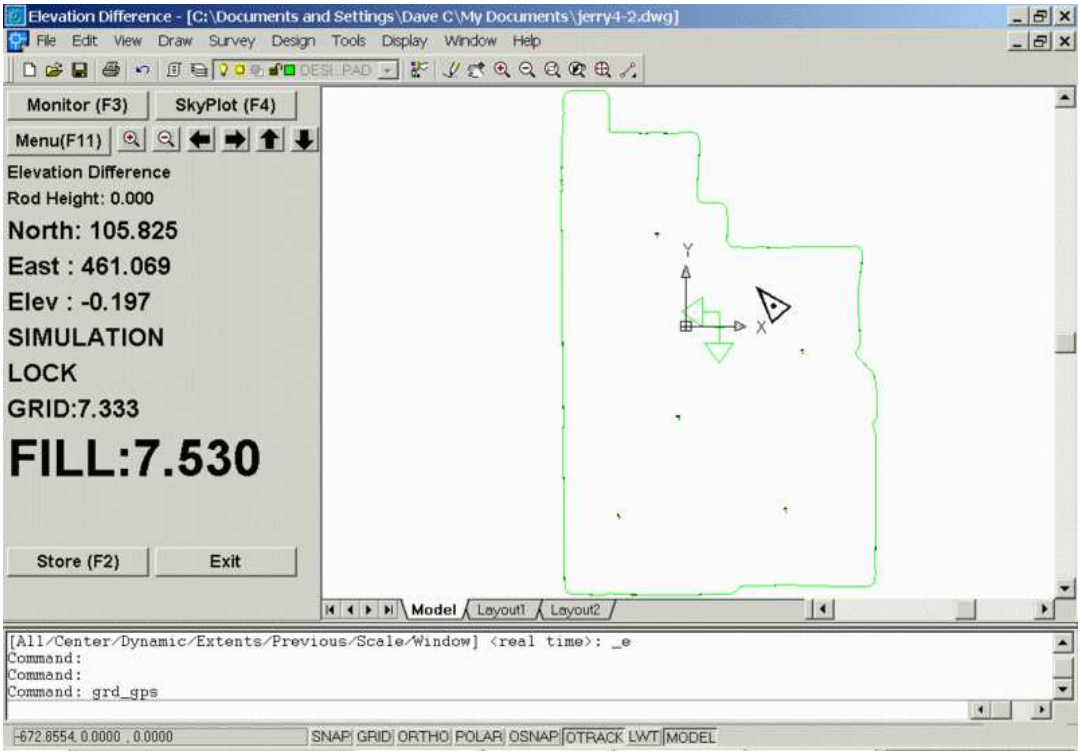


Stakeout Design Surface

This feature reports the cut/fill at each point in the field over which you drive.

Steps

1 Select *Stakeout Design Surface* from the *Tools* menu, and begin to pilot your vehicle. When complete, press *Exit*



Pull-Down Menu Location: Survey
Prerequisite: A field design
Keyboard Command: grd_gps

Design Commands 8

Set Perimeter

Use this command to select the "CLOSED" polyline that defines the outer most limit of the disturbed area. Volume calculation will take place inside this boundary.

Prerequisite: a closed polyline
Keyboard Command: tag_inclu

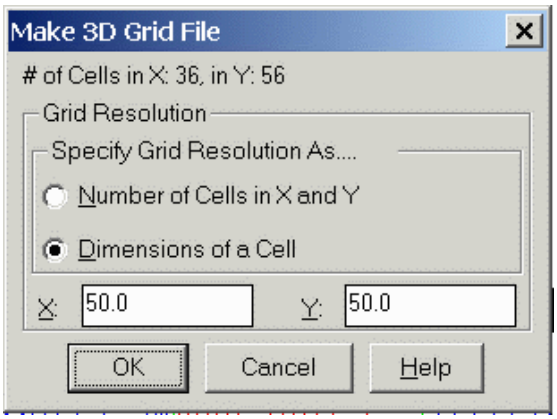
Create Existing Ground Grid

AgStar uses a grid to model the field. The area for the grid is automatically set to cover all the 3D entities in the drawing. Before designing a field, you must set the grid density. For fields with a lot of natural slope variation, the dimensions of the grid cell should be made smaller to ensure high accuracy. However, 50X50 should be dense enough for most fields. Note that the higher the density, the more time it will take to calculate the optimal field design.

Steps

1 Set the toggle to *Dimensions of a Cell*, and specify the desire cell size. 50X50 is recommended. (higher density corresponds to lower cell dimensions)

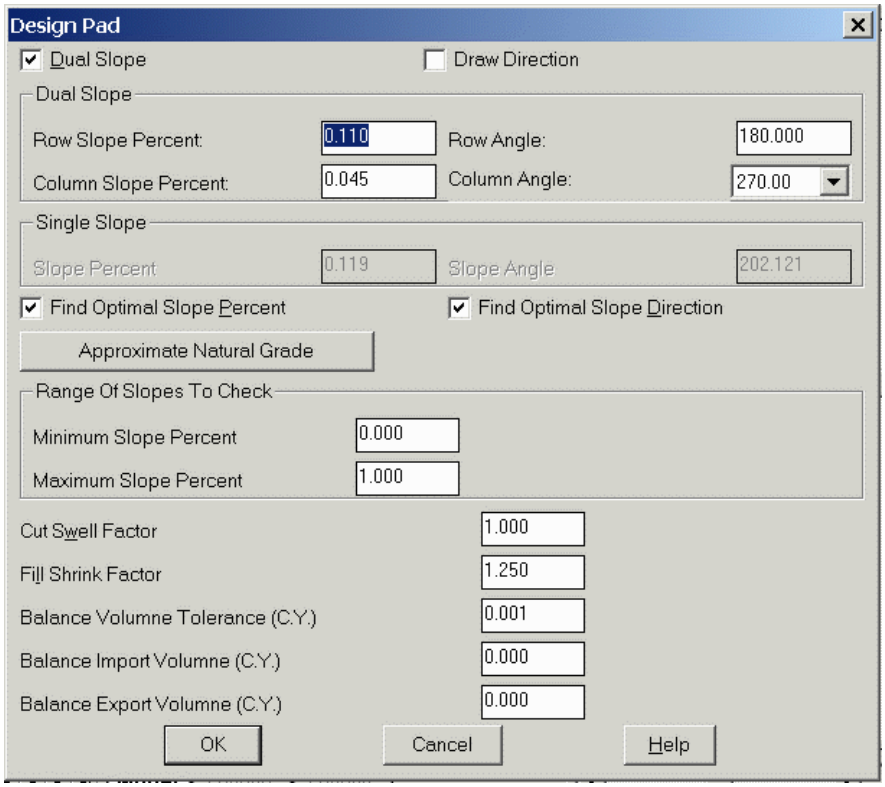
2 Press *Ok.*



Pull-Down Menu Location: Design
Prerequisite: A closed perimeter
Keyboard Command: mk_exgrd

Design Field

This routine allows you either to specify a desired field design, or have Agstar calculate the optimal field design through an iterative process. If the field has been divided into multiple sections, field design can either be performed on a specific section, or on the entire field as a whole.



The design field menu has the following options:

Dual Slope: Check *Dual Slope*, if you would like to specify your slope in terms of rows and columns. Uncheck this option if you would like to specify your slope as a single direction value representing the direction along the fall line. Both methods are equally effective, and the choice between them is merely one of preference.
Draw Direction: Check *Draw Direction* if you would like to specify the fall direction by drawing it with your mouse. Uncheck this option if you would like to enter the fall direction in degrees with respect to North.
Find Optimal Slope Direction / Find Optimal Slope Percent: To calculate a best fit, both *Find Optimal Slope Direction* and *Find Optimal Slope Percent* must be checked. Uncheck both of these if you already know your design parameters.
Approximate Natural Grade: Pressing this button will quickly calculate an approximate best fit for you field. You must press this button before beginning the exact best fit calculation.
Minimum Slope Percent & Maximum Slope Percent: Use these fields to specify a range of slope percents to check during the best fit calculation.
Cut/Fill Ratio: Use the Cut/Fill ratio to define the ratio between the amount the dirt swells when it is removed from the ground, and the amount it shrinks when it is put back into the ground. This value will vary regionally. If you have the local cut swell and fill shrink factors, you can divide the fill shrink by the cut swell to obtain the cut/fill ratio.
Balance Volume Tolerance: Setting this value smaller will cause your cuts and fills to balance more equally, but may also take longer. The recommended value is 0.1.
Balance Import Volume & Balance Export Volume: To import dirt from another field, specify the amount to import in cubic yards in the *Balance Import Volume* field. To export dirt to another field, specify the amount to export in cubic yards in the *Balance Export Volume* field.

Steps If you have split your field into multiple subdivisions, you will now be prompted to select a subdivision to design. Do so with your mouse, or press ESCAPE to override existing subdivisions and create a design for the entire field. If you have not divide up the field, this step is skipped. Now take one of the following steps:
To find the optimal field design, press the *Approximate Natural Grade* button. Verify that both *Find Optimal Slope Direction* and *Find Optimal Slope Percent* are checked, and then press *Ok* to begin the iterative calculation
To specify your own field design, verify that *Find Optimal Slope Percent* is unchecked, enter your desired slope and direction parameters, and then press *Ok*.

Pull-Down Menu Location: Design
Prerequisite: An existing ground grid. Also, a named subdivision, if applicable.
Keyboard Command: slope_pad

Adjust Field Elevation

This command raises or lowers an entire field by a specified amount. Before running this command, a field design must be completed since this command adjusts an existing design. At the first prompt, you can pick inside a design subdivision to adjust, or choose to adjust the entire field. To raise the field, enter a positive elevation difference and to lower the field, enter a negative elevation difference. After the adjustment, the program reports the new quantities.

Prompts

Pick inside subdivision area to select, or press <Enter> to select entire field: *pick point*
Elevation difference: *1*

Pull-Down Menu Location: Design
Prerequisite: A design field
Keyboard Command: dz_pad_agstar

Draw Subdivision Line

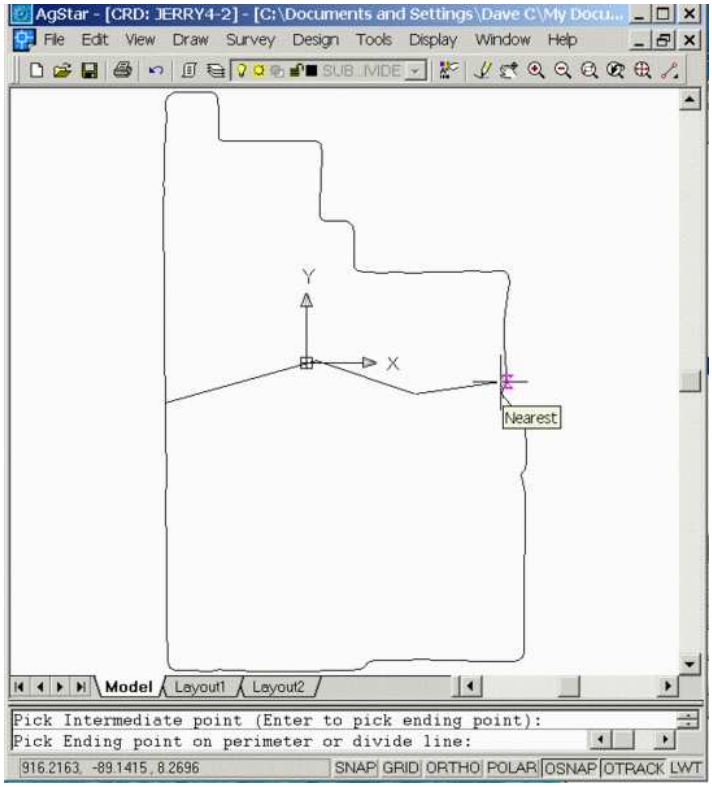
You may wish to divide your field into multiple sub-fields, in order to assign different designs to each area. There are two ways to divide up your field: To draw the subdivision line with your vehicle using GPS, see instructions on the *Survey Subdivision Line* feature. To draw the subdivision line with your mouse, select *Draw Subdivision Line* from the *Design* menu.

Steps

1 Move your mouse to the edge of your field. When two arrows appear, click on the desired start location for the subdivision line.

2 Point and click to draw any number of intermediate points on the subdivision.

3 When you are ready to draw the endpoint, move the mouse to the edge of the field.



4 When two arrows appear, click on the desired endpoint.

5 You will now be prompted to assign names to the two new areas. See instructions on *Assign Subdivision Area Name* for what to do next. Note that you must assign a name to any new section before you can design it.

Pull-Down Menu Location: Design
Prerequisite: A closed perimeter
Keyboard Command: draw_subdivide

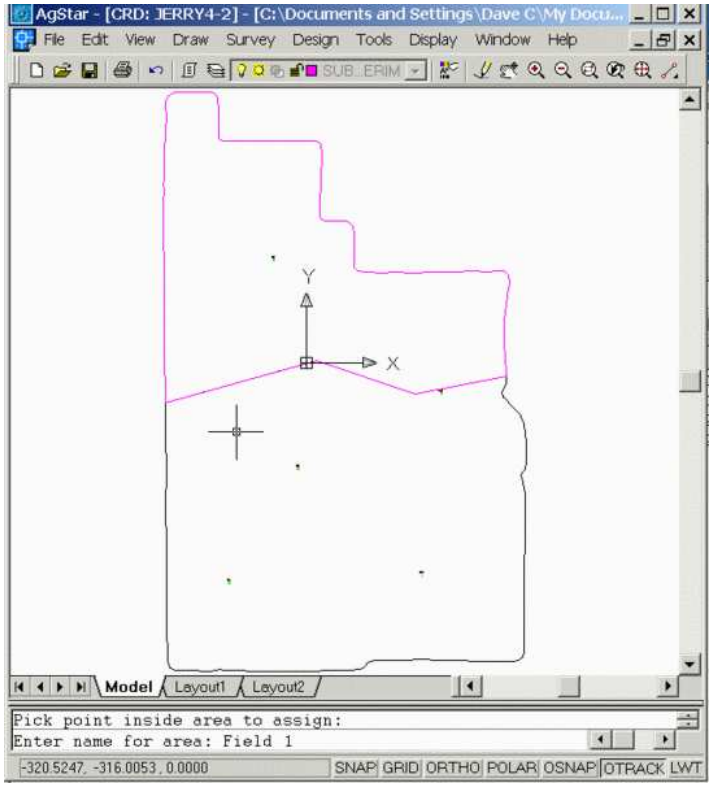
Assign Subdivision Area Name

Before you can design a subdivision, it must first be named.

Steps

1 When prompted to select a subdivision area, click on the subdivision area you wish to name. It should change color.

2 When prompted, type the desired area name and press ENTER.



Pull-Down Menu Location: Design
Prerequisite: Draw or survey subdivision.
Keyboard Command: set_subperim

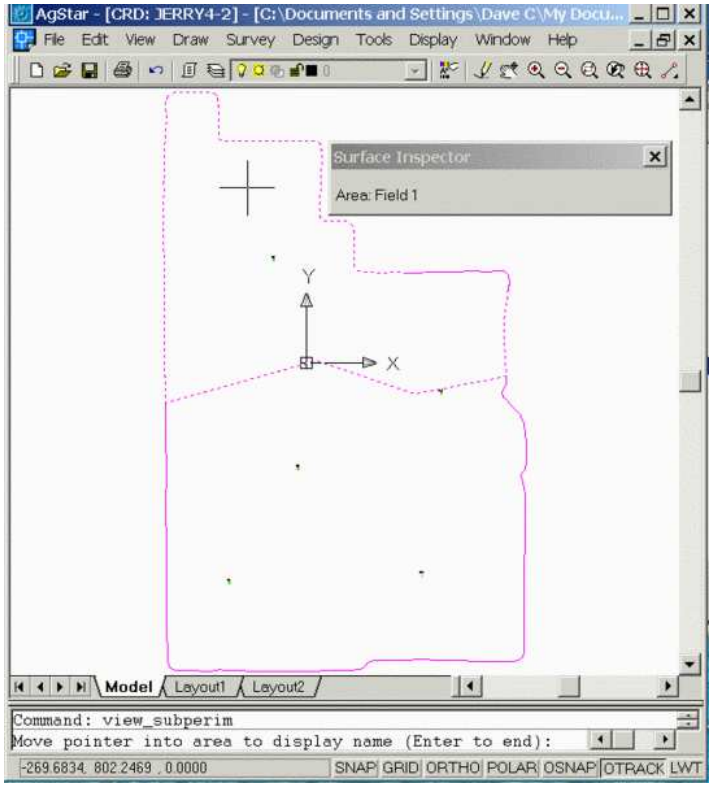
Area Name Inspector

This feature allows you to view the names of each field subarea by dragging the mouse over it.

Steps

1 Drag the mouse over each existing area to view its name.

2 When finished, press ESCAPE.



Pull-Down Menu Location: Design
Prerequisite: The naming of all subdivision areas.
Keyboard Command: view_subperim

Label Area Names

This command draws text labels inside each field subarea. Besides labeling the area name, there is an option to label the area of each subarea. The labels are automatically drawn in the center of each subarea.

Prompts

Text size <10.0>: *press Enter*
Label area size [Yes/<No>]? *press Enter*

Pull-Down Menu Location: Design
Prerequisite: Named subdivision areas
Keyboard Command: label_subperim

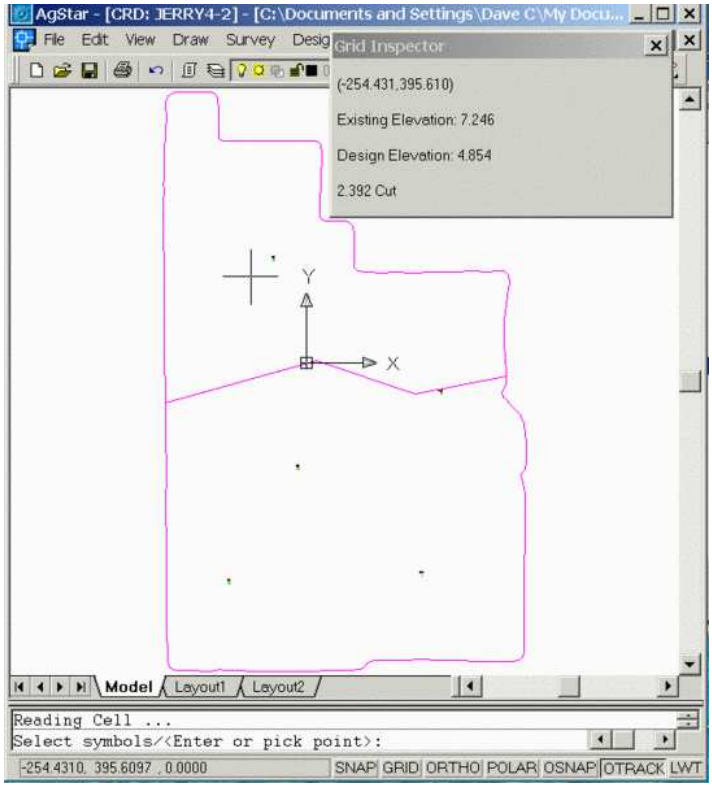
Surface Inspector

Select *Surface Inspector* from the *Design* menu. Drag the mouse to a point to view its existing elevation, design elevation, and cut/fill value. When finished, press ESCAPE. Note that this feature cannot be used until you've run *Design Field*.

Steps

1 Drag the mouse over the field to view the existing elevation, design elevation, and cut/fill values at each point.

2 When finished, press ESCAPE.



Pull-Down Menu Location: Design
Prerequisite: A field design
Keyboard Command: grdvals2

Tools Commands 9

Follow Elevation

This routine will direct you along a chosen elevation using GPS readings. You will need to provide a target elevation and whether the uphill direction is on your left or your right. When you stray out of your chosen elevation tolerance, arrows will appear on the screen, or on an external lightbar, telling you which way to turn to find the target elevation. This feature is useful for designing a field with steps, for example, in planting rice levies.

The following options are available:

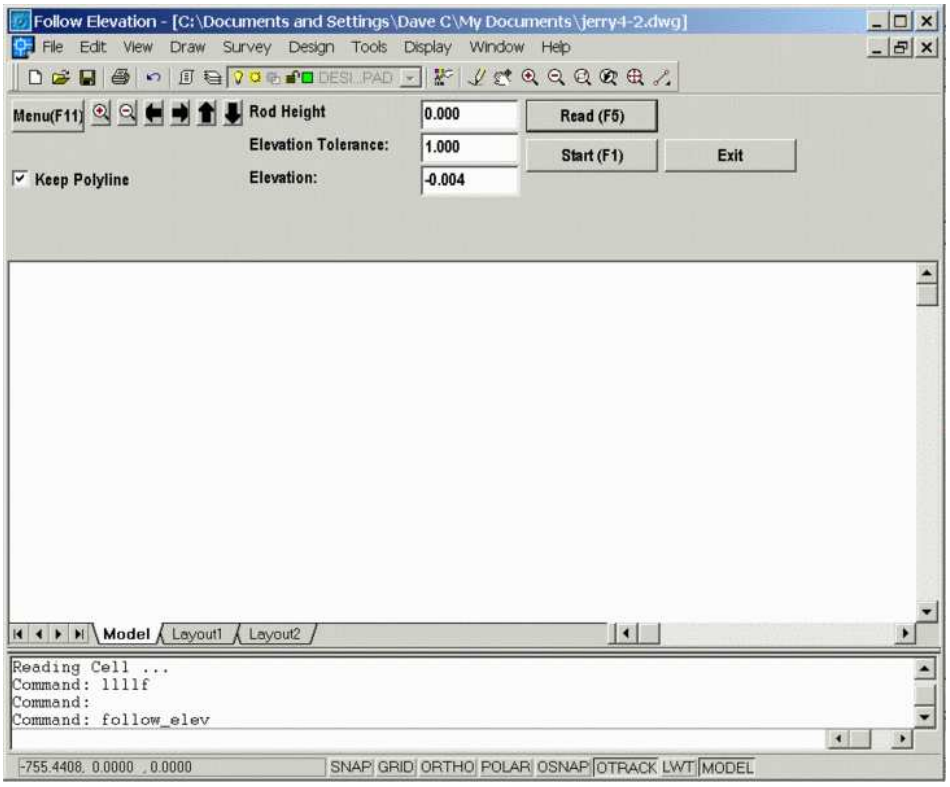
Keep Polyline: Select this options if you would like the path you drive to remain in the drawing.

Elevation tolerance: Set this value to the amount of deviation in elevation you are willing to tolerate.

Elevation: Set this value to the desired elevation you will drive along, or click *Read* to read this value from GPS.

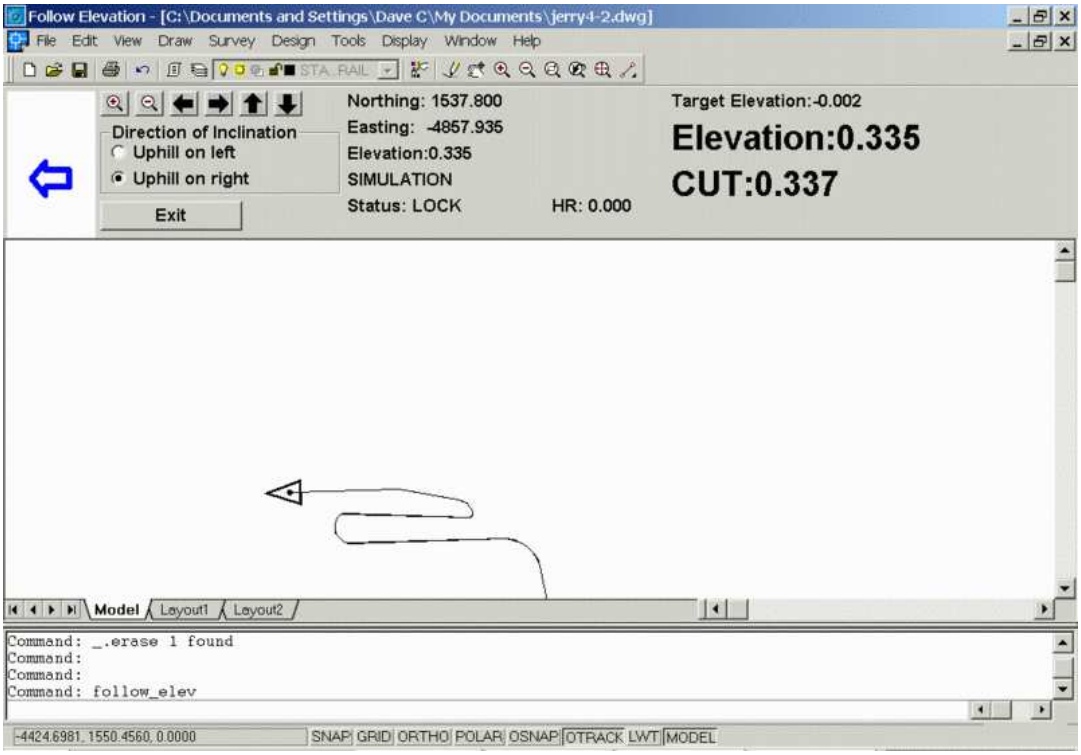
Steps

1 Fill in the desired values for the parameters described above.



2 Click on *Start* to begin.

3 Pilot your vehicle along the desired elevation with the guidance of the left and right arrows on the screen, or using a lightbar. When within tolerance, the arrows become target symbols.



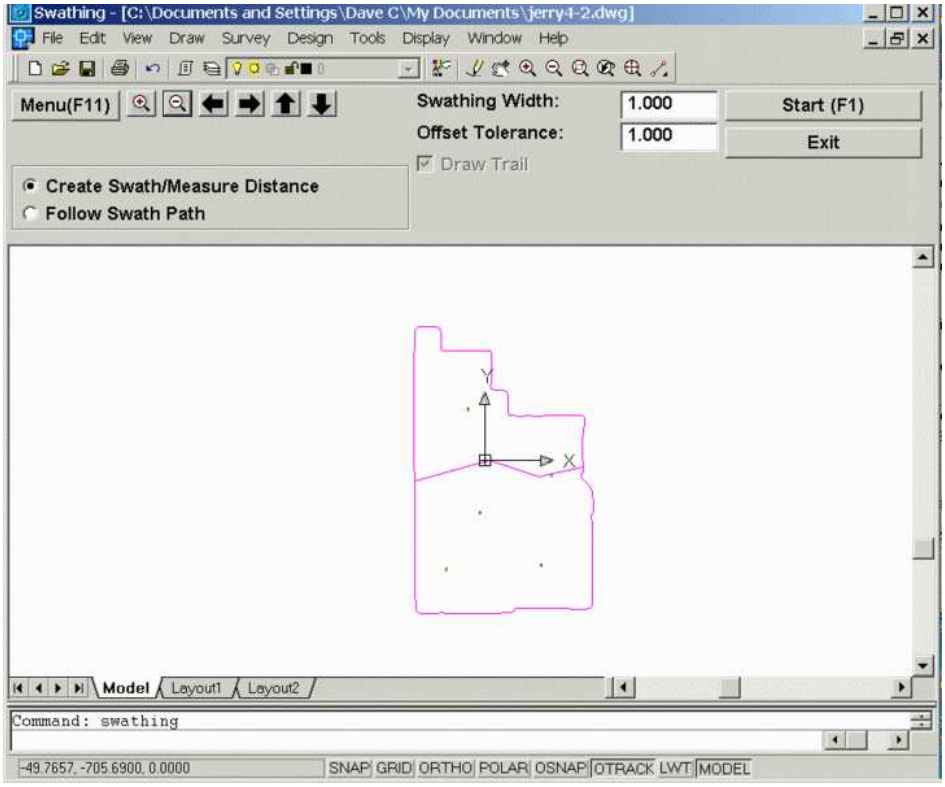
4 Set the direction of inclination as you go (select *Uphill on Left* or *Uphill on Right*). Without knowing which way uphill is, the software will not be able to guide you along the proper course.

5 When complete, click *Exit*.

Pull-Down Menu Location: Tools
Prerequisite: Locked RTK
Keyboard Command: follow_elev

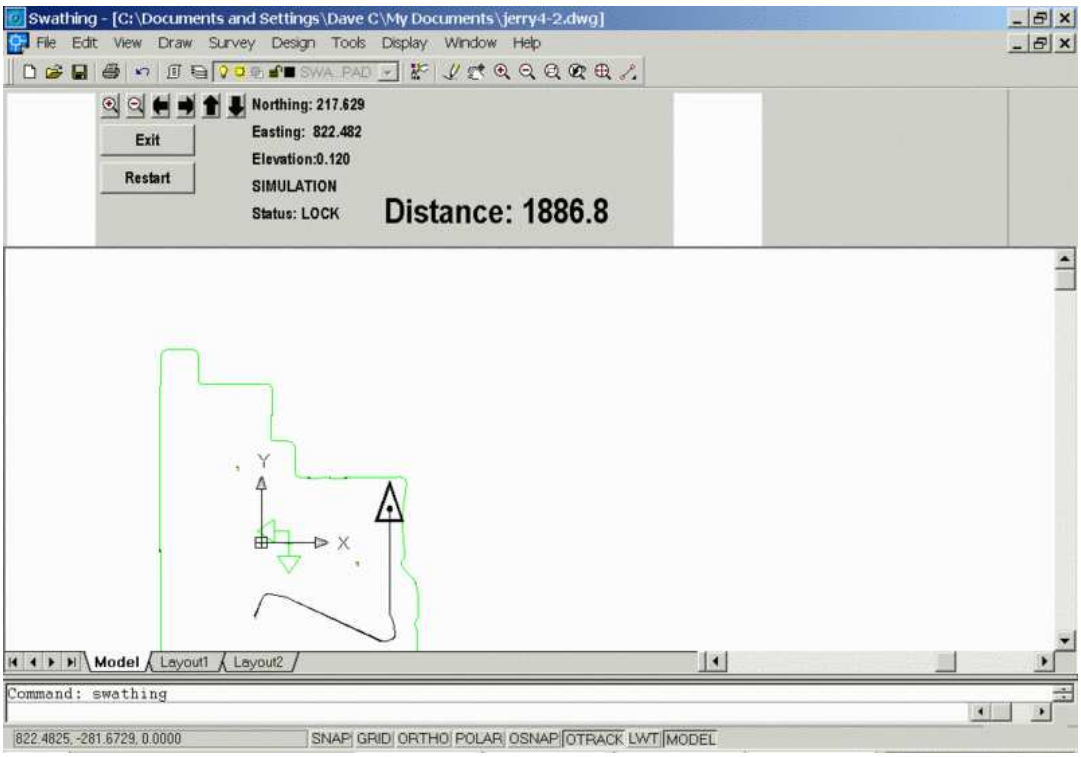
Distance MeasureSwathing

This routine has two uses: To measure a driven distance, and to create or follow a swath of a desired width.



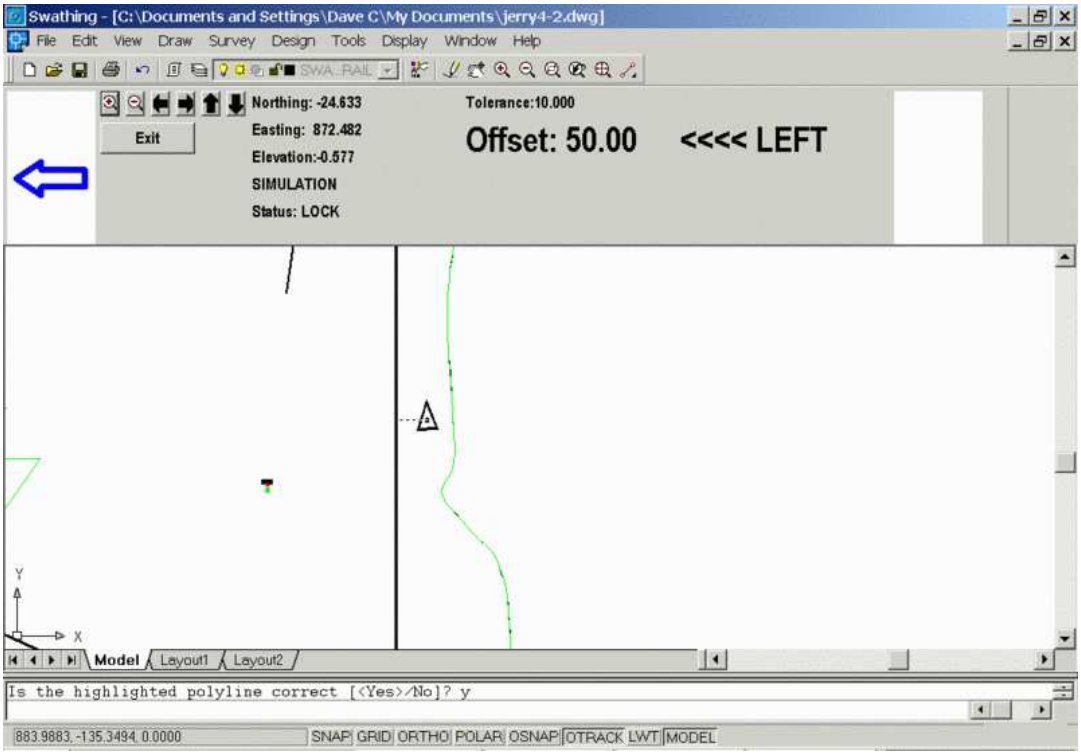
Distance Measure: To measure a distance, set the toggle to *Create Swath/Measure Distance* and press *Start*. The distance will be reported as you go. To restart the measurement, press *Restart*. To exit, press *Exit*. You will then be asked whether or not you would like to save the trail you have driven in the drawing.

Create Swath: Set the toggle to *Create Swath/Measure Distance*. Set the *swathing width* to the desired width of the swath in feet. If you would like to be guided with arrows in such a way that each swath row is a specified distance from the adjacent swath row, set the *swathing tolerance* to the desire value. Press *Start* to begin.



If you want your swaths to be adjacent, on-screen arrows or a connected lightbar will begin to direct you once you have completed a row. Press exit to finish.

Follow Swath: Set the toggle to *Follow Swath.* Set the *swathing tolerance* to the amount of variation in feet you are willing to tolerate from the swath you will follow. If you would like to draw a trail of the path you drive, leave *Draw Swath* checked. Press *Start*. You will be prompted to select the swath you want to follow. Then you will be directed along the chosen swath by on-screen arrows or a connected lightbar. Click *Exit* to stop.



Pull-Down Menu Location: Tools
Prerequisite: None
Keyboard Command: swathing

Distance Measure with Pen

Click on any two locations in the field, and the distance between them will be reported at the command line.

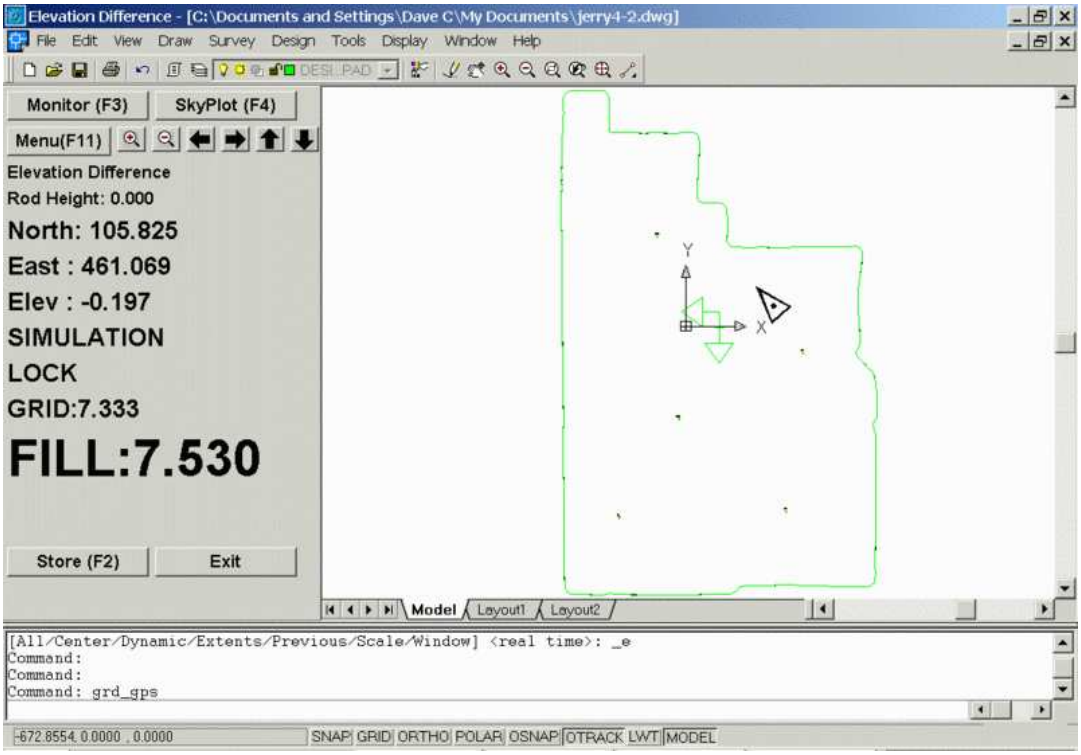
Pull-Down Menu Location: Tools
Prerequisite: None
Keyboard Command: dist

Stakeout Design Surface

This feature reports the cut/fill at each point in the field over which you drive.

Steps

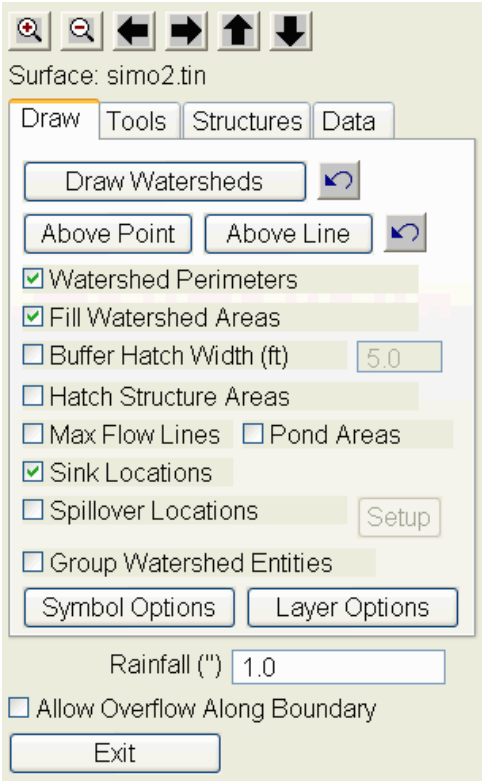
1 Select *Stakeout Design Surface* from the *Tools* menu, and begin to pilot your vehicle. When complete, press *Exit*



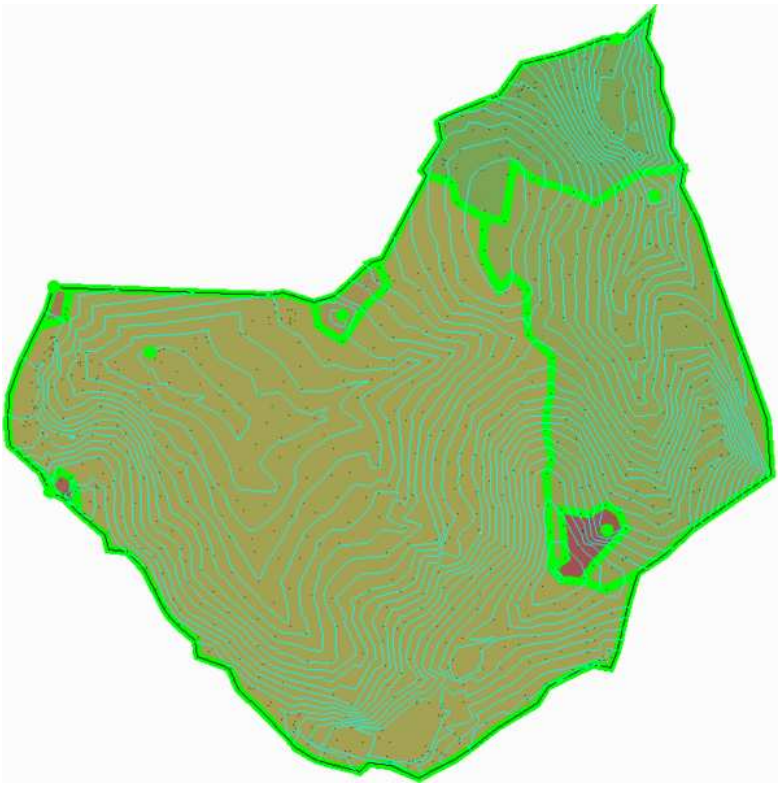
Pull-Down Menu Location: Design
Prerequisite: A field design
Keyboard Command: grd_gps

Watershed Analysis

This command has a collection of tools to analyze the runoff of a surface defined by a triangulation or grid surface file. After selecting the surface file of the surface, the program docks a dialog on the left side of the drawing window. While the Watershed Analysis dialog is running, other AutoCAD and Carlson commands are not available. To zoom or pan the drawing view, use the buttons at the top of the dialog, or use the middle button of a wheel-mouse.



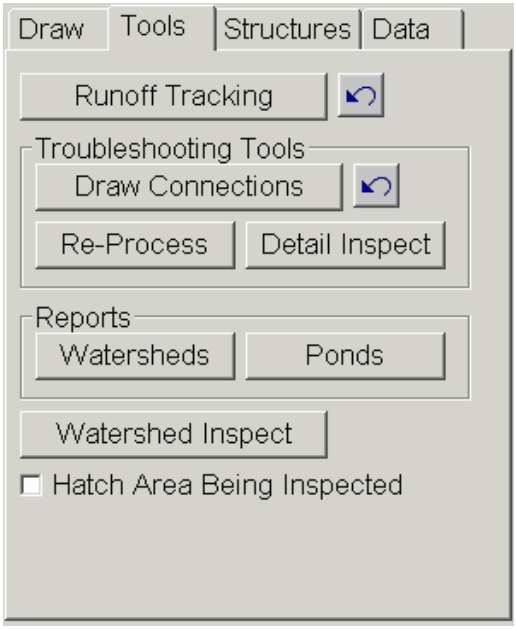
Watershed Analysis calculates the flow connections between the triangles and along the edges of the triangulation. The **Rainfall** amount is used in the processing for figuring the runoff volume to determine when the volume is enough to spillover a local depression in the surface. Besides the Rainfall amount, the runoff coefficients as defined in Define Runoff Layers are also used to calculate the runoff volumes. When the local depression is small enough the runoff will continue through. Otherwise this spot is called a sink for where the runoff stops. The **Round to dZ** is a process option that rounds the elevations of the surface model to simplify the processing. Set this value to zero for no rounding. The **Allow Overflow Along Boundary** option applies to watersheds that have runoff that hits the surface border. This option will check whether this border runoff can spillover and merge with the neighboring watersheds along the border.



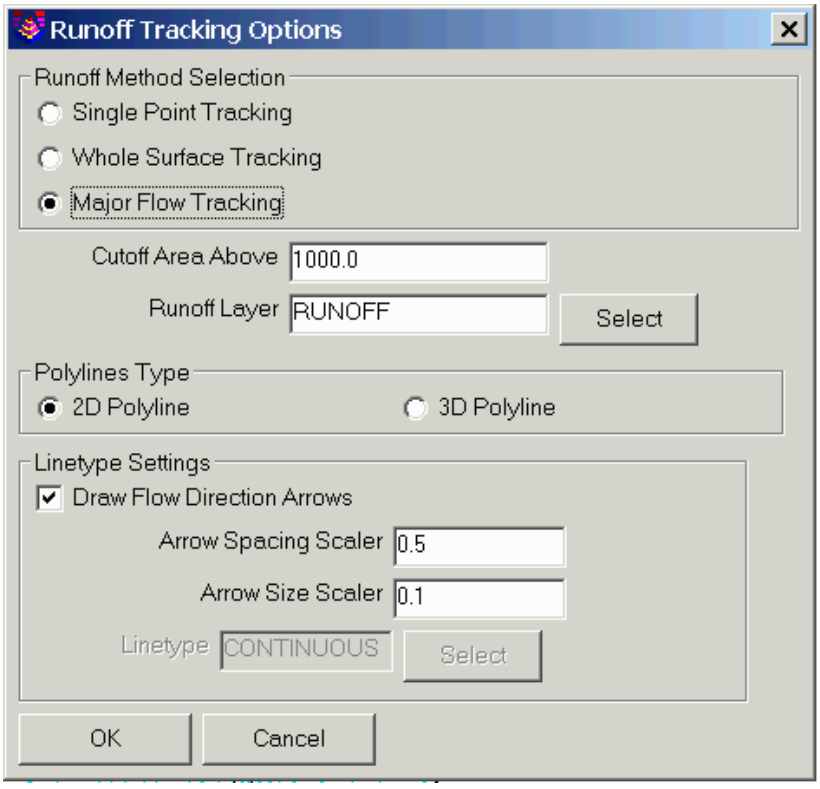
The **Draw Watersheds** function draws the watershed areas using the settings under the Draw tab. The back arrow next to the Draw Watersheds button will erase any previous Draw Watershed entities. The **Watershed Perimeters** option will draw closed polyline perimeters for each watershed area. The **Fill Watershed Areas** option will solid fill hatch each area using different colors. The **Buffer Hatch** option will hatch the perimeters of the watershed areas with the specified width instead of hatching in the entire watershed area. The **Hatch Structure Areas** option will hatch the drainage areas covered by structure inlets defined in the Structures tab. The **Sink Locations** setting draws a symbol at the low point for each drainage area. The **High Point Locations** option draws a triangle symbol at the highest point within each watershed. Typically, this high point will be along the watershed boundary polylines that follow the high points along the ridges between the watersheds. The **Pond Areas** option draws a solid fill hatch in blue for the area covered by the runoff volume of low points. In the example shown, the Fill Watershed Areas and Sink Locations options are active. The **Max Flow Lines** option draws polylines for the longest flow line within each watershed. These longest flow polylines can be used to calculate the time of concentration. The **Spillover Location** option draws symbols at low points within the watershed area that fill up with runoff and spillover on the way to the lowest (sink) location of the watershed. The Setup button allows you to specify criteria for identifying spillover points. These settings include the minimum drainage area, storage volume, drainage volume and ponding depth. These settings allow you to filter out small spillover points (ie a pothole) and only draw the significant ones. The **Group Watershed Entities** option will make AutoCAD groups for the set of entities drawn for each watershed. The **Symbol Options** and **Layer Options** buttons allow you to set the symbols and layers to use for the entities created by Watershed Analysis.

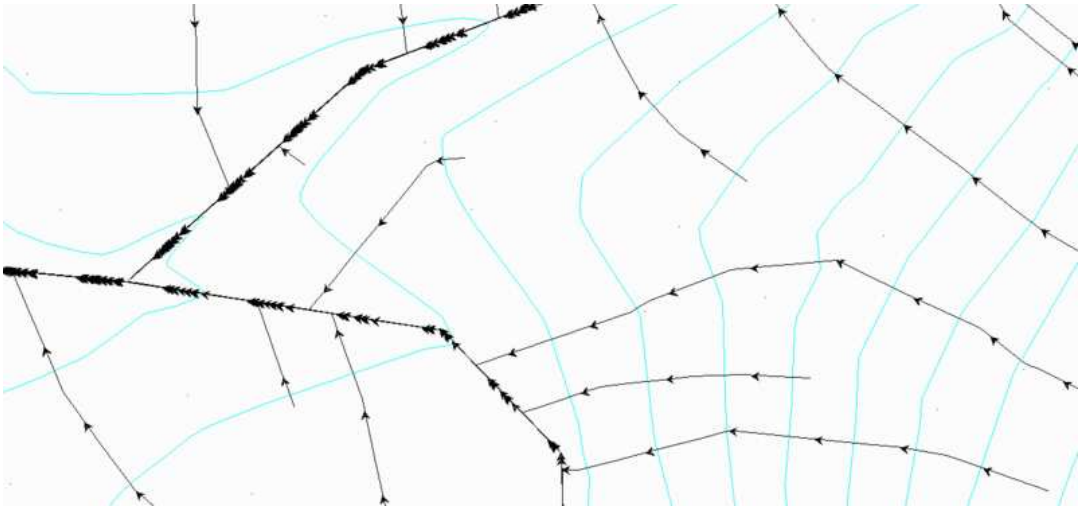
The **Above Point** function reports the watershed data of the current pointer position in real-time as the pointer is moved around. The watershed data is shown in a tooltip next to the pointer position. This data has values for the overall watershed that the position is in including the sink elevation, sink name, drainage area and average slope percent. This data also has values for the watershed above the current point including the drainage area and runoff volume. Plus this data shows the elevation and runoff coefficient at the current point. If the position is picked with the mouse, then the program draws a polyline perimeter for the drainage area above the current point.

The **Above Line** function is similar to Above Point except that you pick two points and the program draws the watershed for all flow that crosses the line between these two points. For example, you can pick points at the left and right banks of a stream to get the drainage area for that stream above these points.

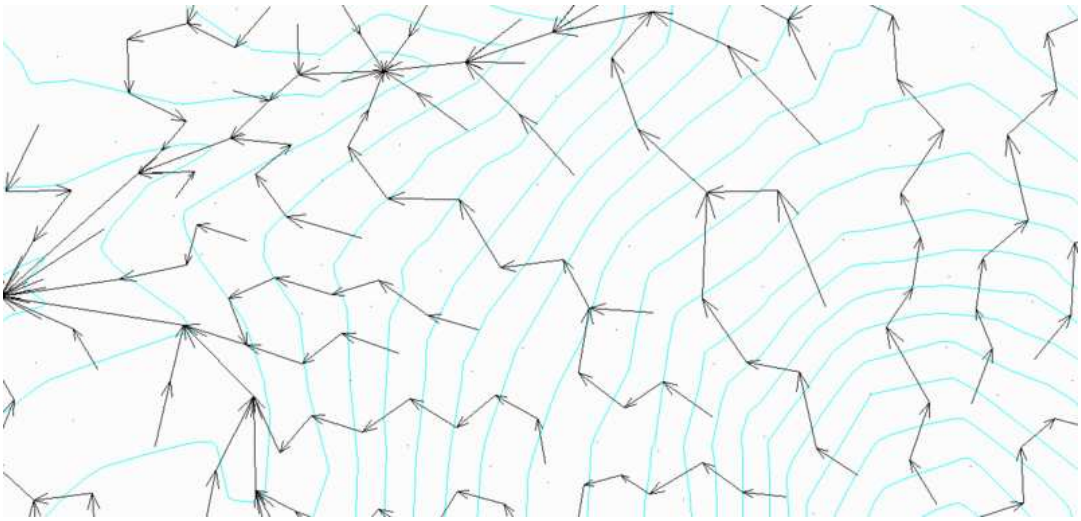


Under the Tools tab there are several analysis routines. The **Runoff Tracking** function draws flow lines that follow the surface. The **Single Point Tracking** method draws the flow lines starting from the picked high points. The **Whole Surface Tracking** method draws a flow line starting from the middle of each triangle in the triangulation. The **Major Flow Tracking** method draws starting in triangles where the drainage area coming into triangle exceeds the specified **Cutoff Area Above** value. The flow lines can be drawn as either 2D or 3D polylines. For 2D polylines, the linetype can be specified or the special linetype with flow direction arrows can be used. This special flow linetype has controls for the size and frequency of the flow arrows.





The **Draw Connections** function draws lines with arrows between the triangles for how the program has determined their flow connections.



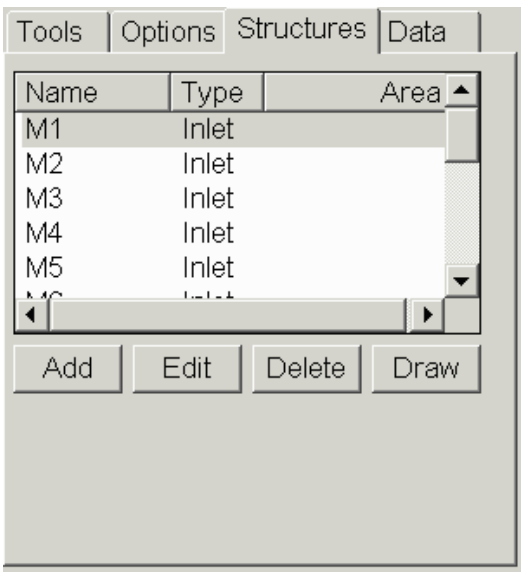
When a triangulation file is processed by Watershed Analysis, some of the flow connection data is stored into the triangulation file to speed up reprocessing. The **Re-Process** function resets this flow connection data to start the flow calculations from scratch.

The **Detail Inspect** function reports flow connection data at the pointer position in real-time as the pointer is moved. This data includes the current position triangle number, connecting flow triangle number, sink node number, watershed name, border elevation, ridge elevation, low elevation, downstream sink number, number of source triangles, number of source nodes, current elevation and spillover elevation.

The **Watershed Inspect** function reports runoff flow data at the pointer position in real-time as the pointer is moved. The runoff data is shown in a tooltip next to the pointer and in the **Data** tab. This data has values for the overall watershed that the position is in including the sink elevation, sink name, drainage area and average slope percent. This data also has values for the watershed above the current point including the drainage area and runoff volume. Plus this data shows the elevation and runoff coefficient at the current point. When the Hatch Area Being Inspected option is active, the watershed area for the current position is hatched during inspection.

The **Watersheds Report** function runs the report formatter to choose which of the watershed parameters to report. The **Ponds Report** function reports the position and depth of each ponding area.

Besides calculating the runoff of the triangulation surface, Watershed Analysis can also process the runoff effects from structures for inlets, storage ponds, culverts and channels. The structures in Watershed Analysis are simply for placement and watershed delineation. These structures do not have design considerations for parameters like pipe size. In the **Structure** tab, there is a list of the structures to apply with the current surface. The list shows the name, type and drainage area for each structure. The Draw function will draw symbols for each structure. The Inlet structures act as sinks in the watershed and capture all the flow that comes to the inlet point. Each inlet is defined by a single point and a name. The Storage Tank structures also act as sinks and are defined by a single point and name. The Culvert structures route the flow from the culvert inlet to the outlet. The culverts are defined by two points for the inlet and outlet and by a name. The Channel structure is the same as the Culvert except that it can have more than two points to define the flow path. The structure data can be stored to a Watershed Structure File (wst) using the **Save** button. The **Load** button can read the structure data from either a wst file or from a sewer network file (.sew).



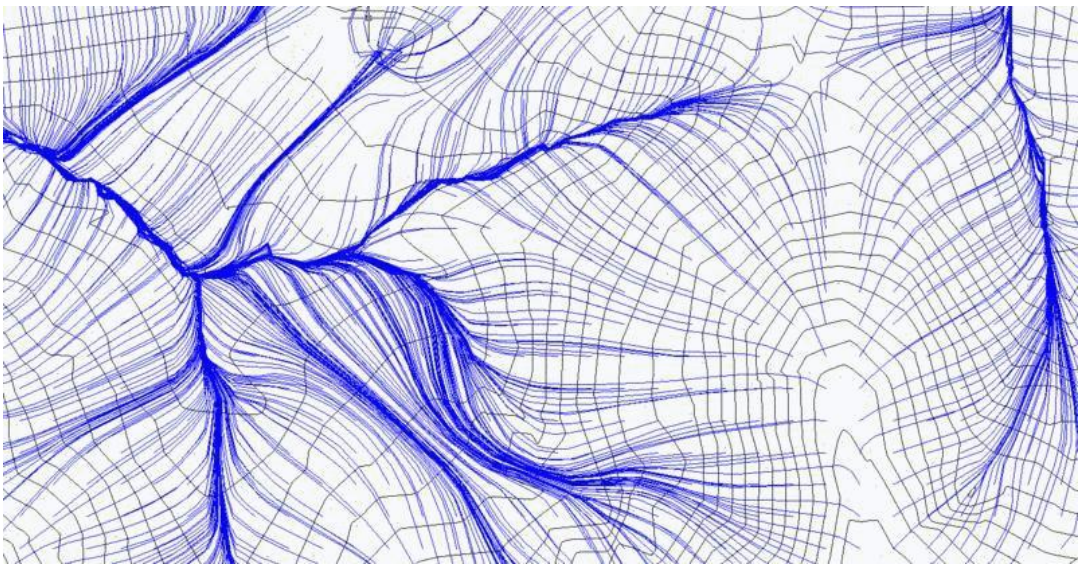
Pulldown Menu Location: Tools
Keyboard Command: watershed
Prerequisite: Triangulation File or Grid File created by Design Field

Run Off Tracking

This command draws 3D polylines starting at user picked points downhill until they reach a local minimum or the end of the grid or TIN. In effect it simulates the path of a rain drop. The program also reports the horizontal and slope distances, average slope, maximum slope, and vertical drop. These values can be used for time of concentration calculations. Runoff tracking is a convenient way to identify distinct watershed areas and is an alternative to the automated Watershed Analysis command.

Prompts

Enter the run off path layer <RUNOFF>: *press Enter*
Select Surface Model dialog box
Choose the grid file or triangulation file that models the surface. If a grid is selected, it will prompt:
Extrapolate grid to full grid size (Yes/<No>)? *Yes* If the limits of the surface data doesn't cover the entire grid area, then the values for the grid cells beyond the data limit must be extrapolated in order to compute slopes in that area. This prompt only appears if there are grid cells without values.
Local pond spillover depth <4.80>: *press Enter* This allows the runoff line to continue past flat or low points in the grid or TIN, by allowing these area to fill up with water, in essence, up to the specified depth, thus letting the runoff polyline continue on.
Draw tracking for all grid cells or pick individuals [All/<Pick>]: *press Enter* Pressing Enter leads to individual picking of runoff tracking lines, while A for All would fill draw runoff polylines starting from each grid cell or each triangulation triangle.
Pick origin of rain drop: *pick a point at the top of the run off polyline*
Pick origin of rain drop (Enter to end): *press Enter*



Pulldown Menu Location: Tools
Keyboard Command: runoff
Prerequisite: A .grd file created by Design Field

Point Defaults

This command sets AgStar point options.

Descriptions: Specify whether you are prompted for a point description when creating points and whether the point descriptions are labeled in the point block.

Elevations: Specify whether you are prompted for a point elevations when creating points and whether the point elevations are labeled in the point block.

Locate on Real Z Axis: When checked, points are located at their actual elevation, otherwise points will be located zero elevation.

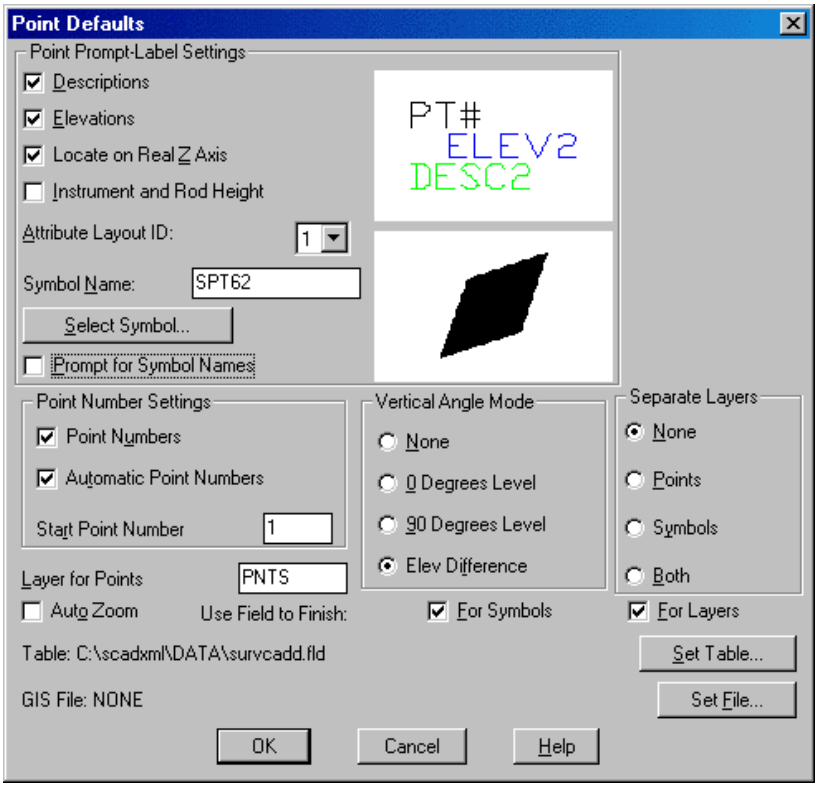
Attribute Layout ID: Controls the location of the point number, elevation and description. These attribute layouts are defined in AutoCAD drawings that are stored in the AgStar SUP directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then open and edit the associated SRVPNO drawing.

Symbol Name: Enter the default symbol name to use. You may also pick the Select Symbol button to select a symbol from the symbol library.

Prompt for Symbol Names: When checked, you will be prompted for each symbol name instead of using the default symbol.

Point Numbers: When this toggle is OFF, no point number will be created and no points will be stored in the coordinate (.CRD) file.

Automatic Point Numbers: When this toggle is OFF, commands that locate a point will prompt for a point number. Otherwise, point numbers are numbered sequentially. If the Start Point Number field is set to 0, no point will be plotted. An exception to this is when you use the *Draw-Locate Points* command and use the Range option, then a point entity is plotted.



The following table illustrates the effects of elevation settings:

Elevations Yes Real Z No

Picked Point Labels point, Prompts for elevation, uses 0 for z coordinate

Point Number Labels point, No Prompt, uses 0 for z coordinate

Elevations Yes Real Z Yes

Picked Point Labels point, Prompts for elevation for z coordinate

Point Number Labels point, No Prompt, uses z coordinate from file

Elevations No Real Z No

Picked Point No Label, No Prompt, uses 0 for z coordinate

Point Number No Label, No Prompt, uses 0 for z coordinate

Elevations No Real Z Yes

Picked Point Labels point, No Prompt, uses z coordinate of picked point

Point Number Labels point, No Prompt, uses z coordinate from file
Start Point Number: Specify the next point number to use.

Vertical Angle Mode: Specify how AgStar should prompt you for vertical angles. None means no prompt. Applies to creating points with commands such as *Traverse*. The vertical angle is used to calculate the point elevation.

Separate Attribute Layers: Specify settings for point attribute layers.

None: The point symbol, point number, elevation and description use the layer names PNTMARK, PNTNO, PNTELEV and PNTDESC.

Points: The point number, elevation and description layers are composed by concatenating the point layer and the string NO, ELEV, and DESC respectively. For example, if the point layer is UTIL then the attribute layers will be UTILNO, UTILELEV and UTILDESC.

Symbols: The point symbol layer is composed by concatenating the point layer and the string MARK. For example, if the point layer is UTIL then the symbol layer will be UTILMARK.

Both: The point symbol, point number, elevation and description layers are composed by concatenating the point layer and the string MARK, NO, ELEV, and DESC respectively. For example, if the point layer is UTIL then the symbol/attribute layers will be UTILMARK, UTILNO, UTILELEV and UTILDESC.

Layer for Points: Specify the layer name for AgStar points.

Auto Zoom: When checked, AutoCAD will perform a Zoom—Center around new points to keep the display centered around current working area. This only applies during commands such as Traverse. This setting is also available in *Configure Survey* under General Settings where it is called Auto Zoom Center for New Points.
Use Field to Finish For: Allows you to use the code definitions from Field to Finish for the point symbols and/or layers when creating new points. For example, when creating a point with description "EP", AgStar would look up "EP" in the Field to Finish table and will use the symbol and layer defined in this code table instead of the symbol and layer set in Point Defaults.

GIS File: This option lets you specify a GIS file to be used when creating new points. The GIS file contains a list of fields to prompt for. For each point that is created, the program will prompt for these fields and store the results to the note file (.not) associated with the current CRD file.

Pull-Down Menu Location: Tools > Points
Prerequisite: None
Keyboard Command: ptsetup

DrawLocate Points

This command inserts either new or existing points into the drawing. New points are created by picking points or by entering northing and easting coordinates. Existing points are placed by entering the points numbers which reference the current coordinate file. You will be prompted to choose a coordinate file if no coordinate file is set current.

Symbol Name: Enter the symbol name to use. You may also pick the Select button to select a symbol from the symbol library.

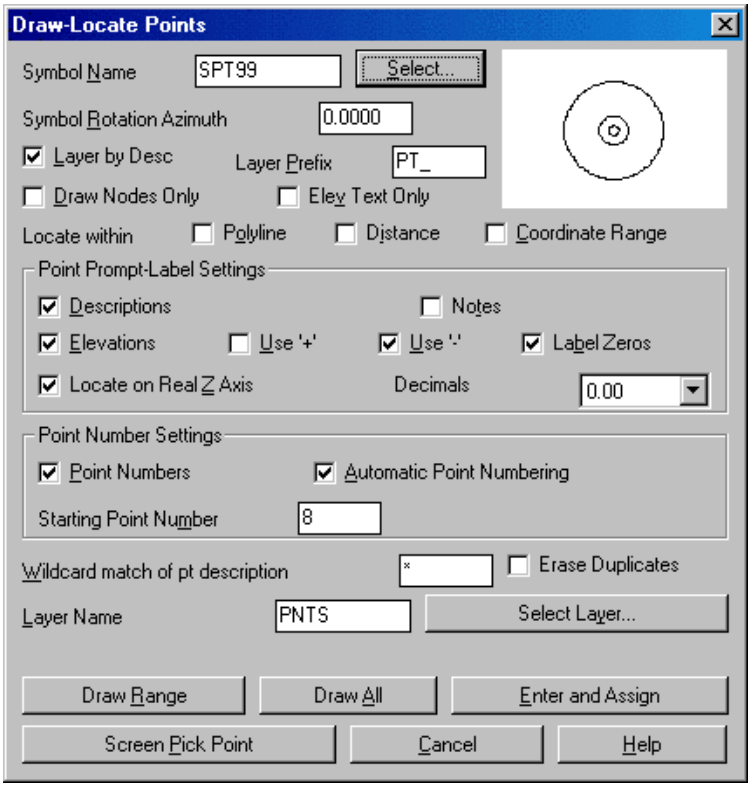
Symbol Rotation Azimuth: Specifies the rotation angle that will be used for the point symbols. This angle is used in a counterclockwise direction relative to the current twist screen .

Layer by Desc: When checked, points are inserted in the layer named by the point description. Using Layer by Desc. organizes the points by description and allows for layer management such as using *Isolate Layers* to show only points on a certain layer. If a character is detected that cannot be used in a layer name, the layer name will be truncated at the invalid character. For example, a point description of "UP=105" would use layer "UP". The Field to Finish command is a more powerful method to insert points by description.

Layer Prefix: This string is added to the beginning of the layer name when using Layer by Dec. For example, a Layer Prefix of "PNT_" and a point with the description "EP" would use the layer "PNT_EP". Layer Prefix is optional. It allows all the point layers to be grouped so that you can select all the point layers by PNT∗.

Draw Nodes Only: When checked, only the AutoCAD POINT entities are inserted and not the point block and symbol. This option is most useful when there are a lot of points to insert because inserting the nodes only is much faster. AgStar routines like Triangulate & Contour and Make 3D Grid File can use these points and do not need the point block and symbol.

Elev Text Only: When checked, the text of the point elevation is drawn without the point block, symbol or node. The decimal place of elevation text is placed at the northing and easting point location.



Locate Within Polyline: When checked, you will be prompted to select a closed polyline. Only the points that are inside this closed polyline will be drawn.

Locate Within Distance: When checked, you will be prompted for a reference point and a distance. Only the points that are within the search distance from the reference point will be drawn.

Locate Within Coordinate Range: When checked, only the points that are within the specified range of northing, easting and elevation will be drawn. The program prompts for the minimum and maximum northing, easting and elevations. These values default to the actual min and max in the CRD file. Then the program prompts for the point number range of points to check.

Descriptions: Specify whether you are prompted for a point description when creating points and whether the point descriptions are labeled in the point block.

Elevations: Specify whether you are prompted for a point elevations when creating points and whether the point elevations are labeled in the point block.

Locate on Real Z Axis: When checked, points are located at their actual elevation, otherwise points will be located zero elevation.

Notes: Works with the note file (.not) associated with the current CRD file. The note file contains unlimited point descriptions in addition to the fixed 32 character point descriptions in the CRD file. When creating points with *Notes* on, the program will prompt for point notes to be stored with the point. When drawing existing point with *Notes* on, any notes for the points are drawn as text entities below the point description.

Use '+': When checked, labels the positive elevations with a leading '+'. Only applies when drawing points from a coordinate file.

Use '-': When checked, labels the negative elevations with a leading '-'. Only applies when drawing points from a coordinate file.

Label Zeros: When checked, points with zero elevation will be labeled when the *Elevations* option is on. Otherwise only points with nonzero elevation will be labeled.

Decimals: Specify the display precision for the elevation labels.

Point Numbers: When this toggle is OFF, no point number will be created and no points will be stored in the coordinate (.CRD) file.

Automatic Point Numbers: When this toggle is OFF, commands that locate a point will prompt for a point number. Otherwise, point numbers are numbered sequentially.

Start Point Number: Specify the next point number to use.

Wildcard Match of pt description: Can be used to filter the points to be drawn. For example, entering "EP" for the wildcard would drawn only points with a description of "EP". An asterisk (∗) is the default setting, it matches any character sequence, meaning no filtering occurs.
Erase Duplicates: When this option is checked, Draw-Locate Points will detect and erase duplicate points on the screen. For example, if you are drawing points 1-10 and point 6 is already in the drawing, point 6 will be erased before points 1-10 are drawn.

Layer Name: Specify the layer name for AgStar points. You may pick the Select Layer button to select from the list of existing layers.

Draw Range: This command will draw a user specified range of points from the current CRD file.

Draw All: This command will draw all the points in the CRD file and then the program will perform a zoom extents the display to show the points.

Enter and Assign: This command can be used to create new points by typing in the point northing and easting.

Screen Pick Point: This command allows you to create points by picking the point coordinate on the screen. For example, you could set the Object Snap to EndPoint and pick the end point of a building polyline to create a point at the building corner.

Prompts

To create a new point.

Locate Point dialog Choose Screen Pick Point.

Pick point to create: *(pick point)*

Enter Point Elevation <500>: *498.43* This prompt only appears if elevation prompting is turned on.

Enter point description: *HUB* This prompt only appears if description prompting is turned on.

To locate a point in the coordinate file (point number 3 in this example).

Locate Point dialog Choose Draw Range.

Point numbers to draw: *3*

PtNo. North(y) East(x) Elev(z) Description

3 4154.28 4147.35 0.00

Point numbers to draw: *1-2.* Locates a range of points. From 1 to 2.

PtNo. North(y) East(x) Elev(z) Description

1 4252.76 4158.32 0.00 RADPT

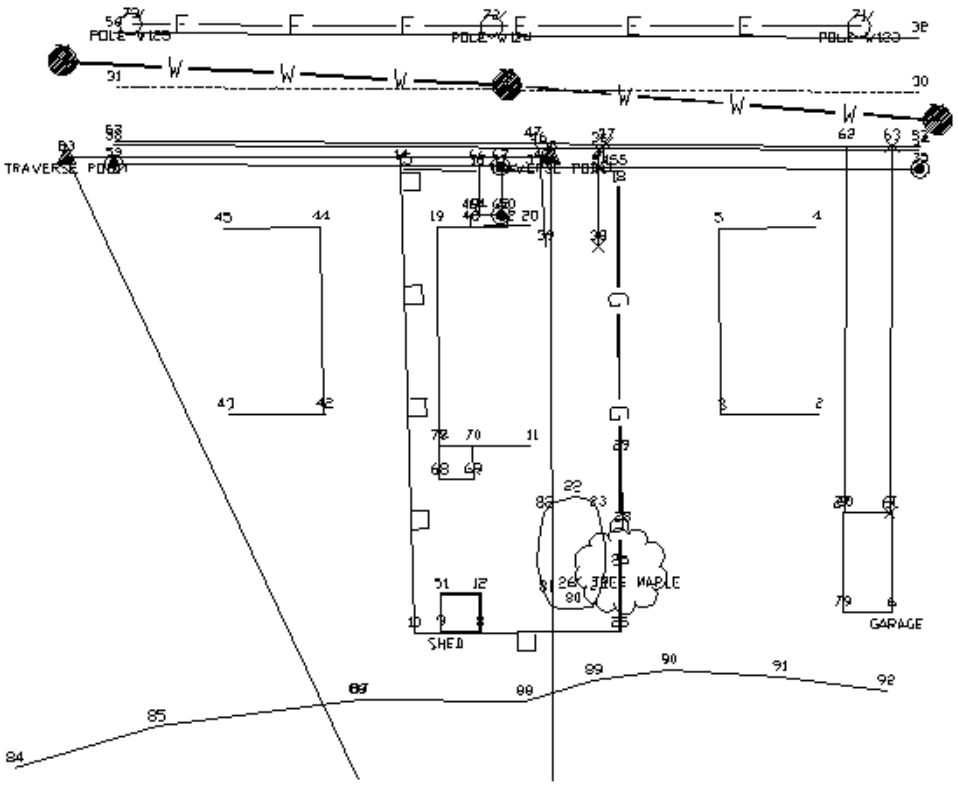
2 4258.11 4059.38 0.00

Point numbers to draw: *Press Enter.* This ends the routine.

Pull-Down Menu Location: Tools
Keyboard Command: LP
Prerequisite: You may want to execute *Drawing Setup* to set the scale and size

Draw Field to Finish

This command turns data collector field notes into a final drawing by matching the descriptions of the field points with user-defined codes. The points are brought into the drawing with attributes defined by the code, including the layer, symbol, size and linetype. Draw Field to Finish also uses an improved coding method.



Example drawing results using the example points and example code definitions

Two files are used in Draw Field to Finish - a coordinate file and a field code definition file. The coordinate file consists of point#, x,y,z points with text description fields. The description fields contain codes for the Draw Field to Finish processing. An ASCII data file can be converted into a coordinate file using the *Import Text/ASCII File* command. The field code definition file defines the layer, symbol, size and other actions to apply with each code. These file names are displayed at the top line of the Draw Field to Finish dialog box.

Draw Field to Finish can translate the field points into Carlson points (also called coordinate geometry points or cogo points) with a symbol, layer, and size defined by the code. The point settings of whether to label the description, point number, and elevation and whether to locate the point at zero or at the real Z can be found in the Additional Draw Options of the Draw Field to Finish dialog box. The *Draw-Locate Points* command has these point settings stored separately in the Point Defaults menu. *Draw-Locate Points* provides a simpler method for

drawing points compared with *Draw Field to Finish*.

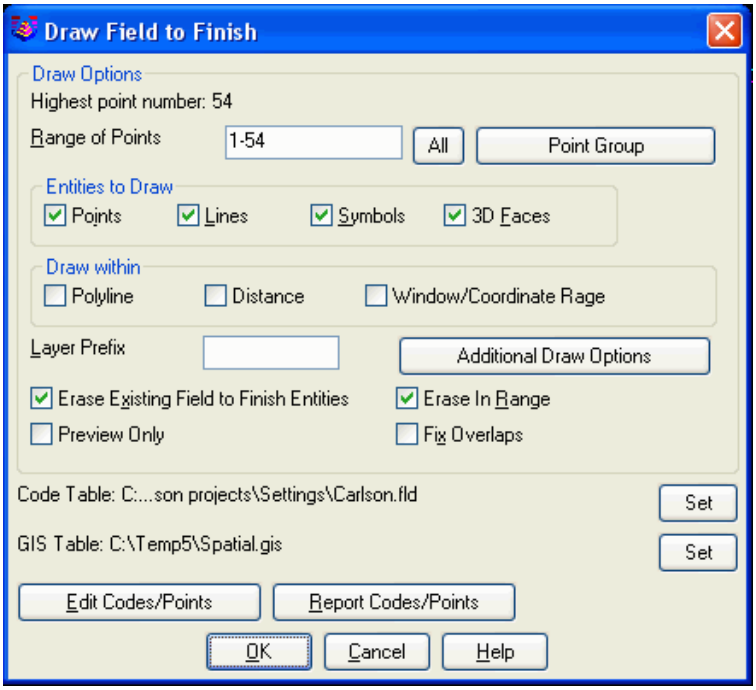
Field-to-Finish will layerize the points and linework according to the code definitions. If the layers to use are not already defined, Field-to-Finish will create the necessary layers and assign different colors. To have the same colors for these layers in all your drawings, define the layers in the prototype drawing. The prototype drawing is the default drawing that is loaded whenever a new drawing is created. To define layers in the prototype drawing, save your current drawing and then start a new drawing with the New command. Don't give the new drawing a name, just click OK. Then define the layers as desired with the *Layer* command. When you are done creating layers, use the Save As command and change to Drawing Template (.DWT) under Save as Type. The default drawing template that is used is named Carlson12.DWT. This template name will correspond to the version of AutoCAD that is being used. You can overwrite this default template or make a new drawing template. If you make a new one, you may want to edit the Carlson icon to use the new one. To edit the icon, highlight the icon with one click and then click the right mouse button. Choose Properties and then Shortcut and change the drawing template name.

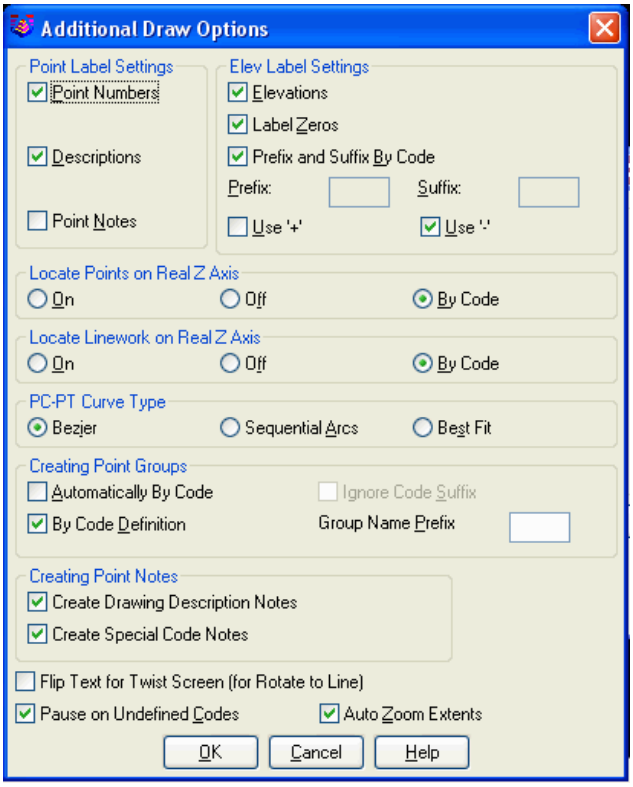
There are two different methods for connecting linework. One method creates line work by connecting points with the same code. The linetype is defined by the code as either points only (no line work), lines, 2D polylines, both 2D and 3D polylines, or 3D polylines (breaklines). Distinct lines with the same code are defined by adding a group number to the end of the code name in the data file. With this method, all points with the description CODE1 will be one line while points with CODE2 will be another line. Both CODE1 and CODE2 use the definition for CODE. For example, the code EP could be a code for edge of pavement that is to be connected as 3D polylines. If there are two separate edge of pavement lines on the left and right sides of a road, all the points for the left side could have the description EP1 and the points on the right side could be EP2.

The second method is the PointCAD format. This method also connects points with the same code. The difference is that instead of using a number after the code for distinct lines, you use the same code with an additional code for starting and ending the line. For example, +0 is used to start a line and -0 to end. So the coding for a segment of edge of pavement could be EP+0, EP, EP, EP-0. Another special code that has been added to Field to Finish is +7, -7. This 7 code will use the linetype definition of line, 2D polyline or 3D polyline defined by the Draw Field to Finish code. For example, if EP is defined as a 3D polyline, then the coding EP+7, EP, EP, EP-7 will create a 3D polyline. Otherwise codes like +0, -0, which is defined as start and end line, will draw EP as a line. Other PointCAD special codes are: +4 starts a curved 2D polyline, ∗4 starts a closed curved 2D polyline, +1 begins a 3-point arc, +5 starts a 3D polyline, ∗5 starts a closed 3D polyline, +6 starts a 2D polyline, ∗6 starts a closed 2D polyline, +7 starts a line whose type is specified by the field code definition, -05 starts a curved 3D polyline section, -50 ends that section, +8 starts a 2D and 3D polyline combination, ∗8 starts a closed 2D and 3D polyline combination, -08 starts a 2D and 3D polyline combination curved section, -80 ends that section. //, followed by a field code, concatenates that field code's description on to the point's description. For example, OAK//04 might become LIVE OAK TREE 4" if the field code OAK translates to LIVE OAK TREE and the field code 04 translates to 4".

The advantage to the PointCAD method is that you don't have to keep track of line numbers. For example, if you are surveying 50 curb lines, the first method would require you to use 50 distinct curb numbers. The advantage to the first method is that you don't have to use the start and end codes. Also the Nearest Found connection option applies to the first method.

Draw





Range of Points: Specify the range of points to draw.

Point Group: Specify the point group(s) to process.

Entities To Draw: The Points option draws only the points and point attributes. The Lines option draws only the linework and the Symbols draws only the symbols. Any combination of these options can be processed as well as individual processing of each entity.

Draw Within: These options are methods to filter the points to draw. The Polyline method prompts for a closed polyline and only draws points inside this polyline. The Distance method uses a specified center point and distance to only draw points within this circle. The Window/Coordinate Range prompts for lower left and upper right points

to define the rectangular area to draw points.

Point Label Settings: Specify whether you want Draw Field to Finish to label the Point Numbers, Descriptions, and/or Points Notes which are contained in the note (.NOT) file that is associated with the coordinate (.CRD) file.

Elevation Label Settings: Specify the elevation labeling options. The Label Zeros option will label the elevations of points with z=0. Use Parentheses will place parenthesis around the elevation text. Use '+' and Use '-' will place the appropriate symbol in front of the elevation.

Locate Points on Real Z Axis: Choose between locating all the points at real Z elevation, all at zero elevation or to use the real Z setting as defined in the individual codes.

PC-PT Curve Type: Sets the method for drawing curves with more than 3 points. The Bezier option draws a smooth polyline through all the curve points. The Sequential Arcs method draws multiple arcs with arc end points at each of the curve points. These arcs are tangent to the preceding line segment. The Best Fit method creates a single best-fit curve for all the curve points between the PC and PT.

Layer Prefix: Optional layer prefix added to all entities drawn with Draw Field to Finish.

Erase Existing Draw Field to Finish Entities: When checked, this option will erase from the drawing any old entities created by previous Field-To-Finish runs before drawing the new entities.

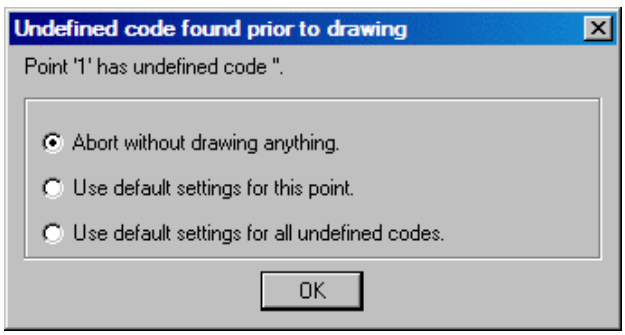
In Range: This option only erases and redraws those Draw Field to Finish entities that are within the specified range of points to process.

Creating Point Groups: Point Groups can be created in one or two different ways. Each field code definition can specify Point Group(s) that all point numbers that use that code will be added to. Multiple field codes can use the same Point Group name. Check the By Code Definition checkbox for that option. The second method is to automatically create Point Groups for each code that is processed. Check the Automatically By Code checkbox for that option. Ignore Code Suffix, if checked, will cause the codes to be considered after removing the numeric suffix. For example, points with the EP10 and EP11 codes will both be automatically added to the Point Group named EP. No matter how the Point Group is created, the Group Name Prefix can be used to add a prefix to the group name. Note: if the Point Group already exists, it will be erased first before being created again by either of these two methods.

Creating Point Notes: These options append point notes to the coordinate file data for some of the data fields processed by Field-to-Finish. These notes can then be used by other commands like List Points to report these fields. For example, this enables List Points to report both the point coordinate file description as well as the point drawing description as generated by Field-to-Finish.

Flip Text for Twist Screen: This option will rotate the point labels and symbol by 180 degrees when needed to make them right-side up readable relative to the current twist screen drawing view. This option applies to the Rotate To Line and Rotate special code (ROT).

Pause on Undefined Codes: When checked, Draw Field to Finish will pause if it encounters a description that is not defined in the code table.



Abort without drawing anything: This stops the command. Run Draw Field to Finish again to correct the code table.

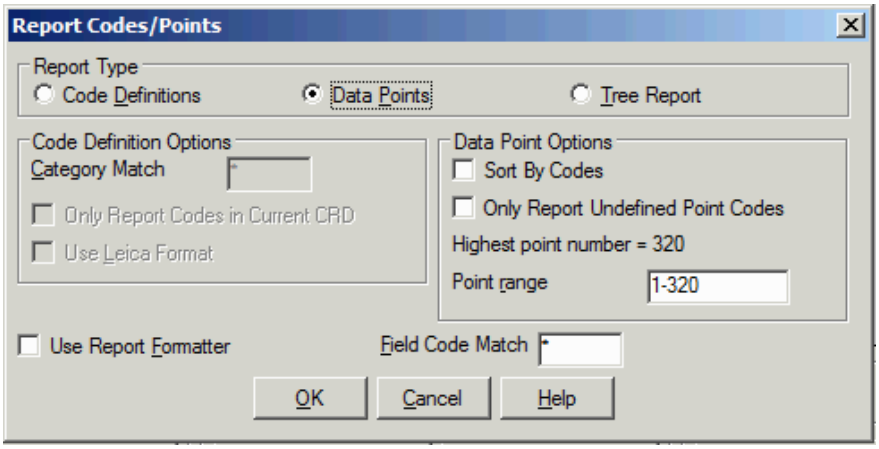
Use the default settings for this point: This option draws a point in the "MISC" layer with no linework. To set your own default, define a code called "SC_DFLT".

Use default settings for all undefined codes: This option will draw all undefined codes in the "MISC" layer by default or a user specified layer as defined in the "SC_DFLT" code. A good way to check the data file for unmatched descriptions is to use the Print Table command and choose the Data Points and Distinct Code options. This command will print the different codes in the data file and identify any undefined codes.

Preview Only: When checked, this option will temporarily draw the points and linework and allow you to review it with zoom and pan.

Auto Zoom Extents: When checked, this will force a zoom extents after Draw Field to Finish is done.

Report Codes/Points: This routine prints the code table or the data file to the screen, file, or printer. A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.

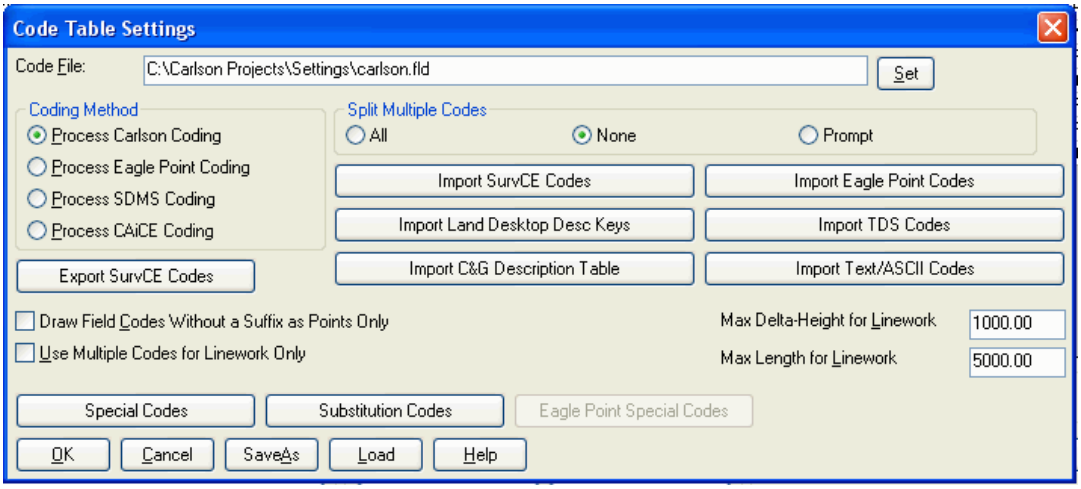


Edit Codes / Points: The Field to Finish dialog box allows you to load the coordinate and field code definition files, view and edit the code definitions, view and edit the coordinate file, view reports, and then return to the Draw Field to Finish dialog box to process the files. The top section displays the code definitions. The bottom section has three columns of functions each pertaining to controls for different elements of the command. The **Code Table** section provides controls for settings, sorting and reporting of codes. The **Code Definitions** section provides tools for the creation and editing of codes. The **Coordinate File** section provides controls for coordinate files and points. It also contains the Draw controls which starts the processing of the data using Draw Field to Finish.

The code table editor has a list of categories and a spreadsheet of codes. The spreadsheet shows the codes for the currently highlighted category. The category toolbar buttons allow you to add, remove, edit the names and change the order of the categories. There are two fixed categories. The Unassigned category shows any codes with blank categories. The All category shows all the codes. You can control which fields are visible in the spreadsheet by using the Column Options button. You can make edits to the fields in the spreadsheet or highlight a row and pick the Edit button to bring up a dialog to edit the code.

Code Table

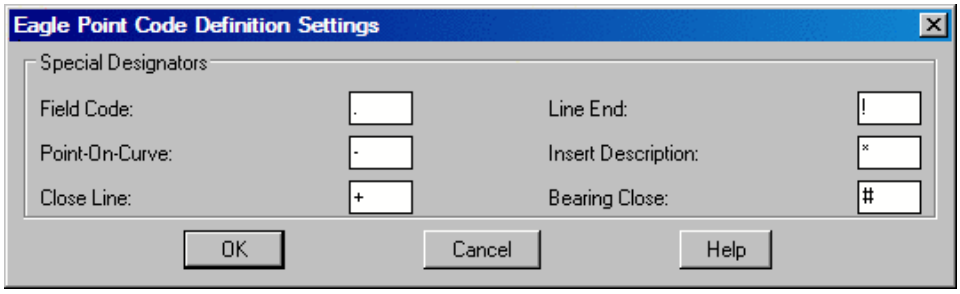
Code Table Settings: These options provide tools for defining the coding method to be used for processing of the point data. Various import tools allow for the importing of codes from different software packages. Controls for handling multiple codes are located on this dialog. All special codes can be replaced to other characters defined by the user. The special codes are listed and edited on this dialog.



Set: Choose this button to specify a new code table. The name of the current table is shown in the field to the right of this button.

Process Carlson Coding: When checked, this option interprets and processes coordinate files based upon the Carlson Coding method and data collection method.

Process Eagle Point Coding: When checked, coordinate files are processed based on the Eagle Point Data Collection method. When selected the ***Eagle Point Codes*** button becomes available for selection and displays the following dialog. This dialog allows for customization of the eagle point special designators.



Currently the supported designators include, "Field Code", "Point-On-Curve", "Close Line", "Line End", "Insert Description" and "Bearing Close". Also supported is the ability to recognize overwriting of descriptions just as Eagle Point does by using the space separator instead of the "Insert Description" designator. Examples of supported coding are as follows:

.TC Places a node and or line per the field code library.
TC Places a node and or line per the field code library.
-TC Specifies a point on a curve.
TC- Specifies a point on a curve.
..TC Stops the line.
TC! Stops the line.
.TC+ Closes the line back to the starting point.
TC+ Closes the line back to the starting point.
.TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.
TC# Typically coded on the third corner of a rectangle to close the figure with having to locate the fourth corner.
WV.W1 Places a node as specified by the code "WV" in the field code library and then begins a line as specified by code "W" in the field code library.
.TC.EP.FL Results in three lines coming together.
TC1.TC2.TC3 Results in three lines coming together. All three lines are specified by the definition of the single code "TC" in the field code library.
TC.TC1 When used in conjunction with the "Draw Field Codes Without a Suffix as Points Only" toggle, "TC" will be recognized as the node and "TC1" will be recognized as the line so that if the code "TC" in the field code library

is defined as a polyline, line or 3D polyline, duplicate lines will not be unintentionally placed when this shot only pertains to a single element. Keep in mind that all line work must have a numeric suffix when using this toggle.
TREE * OAK Result on screen would be: TREE OAK
***TREE OAK** * Result on screen would be: OAK TREE
TREE OAK Result on screen would be: OAK
TC1!.TC2-.VLT6# Stops "TC1", continues "TC2" as a point on a curve and closes VLT6 as a rectangle using the "Bearing Close" code.

Note: The use of the "Use Multiple Codes for Linework Only" toggle is recommended when using Eagle Point Coding.

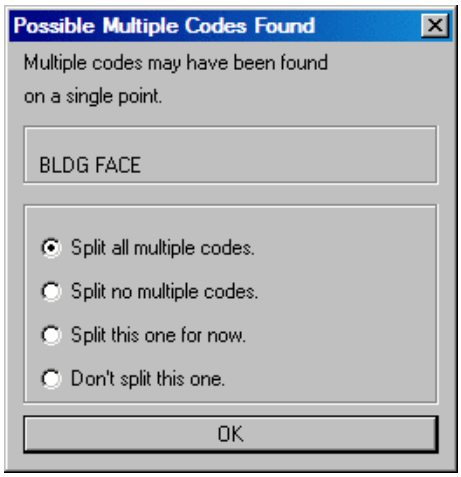
Process CAiCE Coding: When checked, coordinate files are processed based on the CAiCE Data Collection method. Examples of supported coding are as follows:
169 is just the code 169.
145C10 is the code 145 and line #10.
169C25C is the code 169, line #25, and the point is on a curve.
172C12B is the code 172, line #12, and this point closes the line.

Process SDMS Coding: This option processes coordinate files based upon SDMS coding method. When active, the program will prompt for an SDMS .PRJ file to process.

Split Multiple Codes:

Multiple codes are defined by including each code in the point description field separated by a space. A single data point can be used in different lines by assigning it multiple codes. For instance, a point might be part of both a curb line and a driveway line with a description of "CURB DRW". Field-to-Finish uses spaces as the delimiter for multiple codes. You should avoid spaces in the descriptions except for where multiple codes are intended or after the "/" character. For example, a code for light post should not be "LGT POST" but instead should be "LGTPOST".

There are three options for the handling of multiple codes when encountered. The **All** option will split all multiple codes and process each code based upon their code definition. When **None** is select both codes will be processed based upon their code definition. If the **Prompt** option is checked on, when Field-to-Finish detects multiple codes on a point the following dialog will be displayed with options for handling the codes.



Import Land Desktop Desc Key: This option imports and converts a Land Desktop Description Key into a Carlson Draw Field to Finish (fld) code definition file. The Land Desktop Description Key file is a mdb file and is found in the Land Desktop Project file path. It is located in the under the COGO/DescKey directory.

Import TDS Codes: This option imports TDS codes into the Carlson Field to Finish (fld) code definition file.

Import Trimble Codes: This option imports Trimble .FXL file codes into the Carlson Field to Finish (fld) code definition file.

Import Eagle Point Codes: This option imports Eagle Point codes into the Carlson Field to Finish (fld) code definition file.

Import C&G Description Table: This option imports C&G code tables (tbl) into the Carlson Field to Finish (fld) code definition file.

Import Text/ASCII Codes: This option imports code definitions from a user-defined format. Each row in the text file should represent one code. The program will prompt for the delimiter (ie. comma separated) that is used in the text file and then for the field type for each of the columns (ie. "Layer" or "Description").

Import GIS Feature Codes: This option imports features in a .GIS file from Define GIS Features into F2F codes.

Import SurvCE Codes: This option imports a SurvCE Feature Code List (fcl) into a Carlson Field to Finish (fld) code definition file.

Export SurvCE Codes: This option creates a SurvCE Feature Code List (fcl) from the current a Carlson Field to Finish (fld) code definition file.

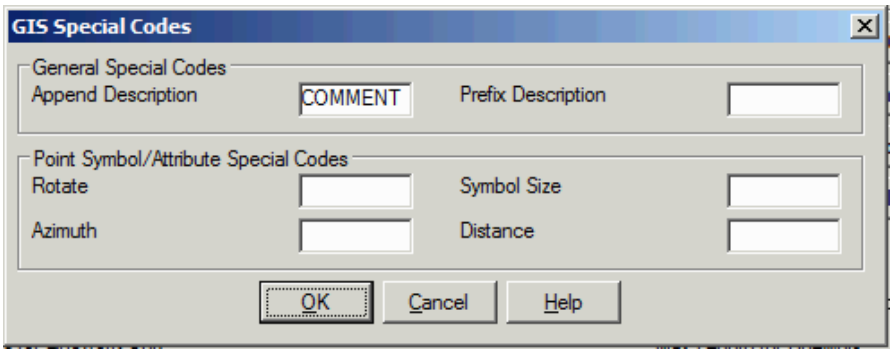
Draw Field Codes Without a Suffix as Points Only: This option is useful for when wanting to use a field code sometimes for linework and sometimes for just points but it is preferred to number the lines rather than using start and stop codes. For example, if the field code EP is defined to use the Line Entity type, then EP25 will be drawn as a Line, however if just EP is used, no linework will connect to that COGO point.

Use Multiple Codes for Linework Only: When checked, and multiple codes are detected, only linework will be drawn for the secondary codes. Points are only created based on the primary code. If you want symbols for all multiple codes, then this setting should not be checked.

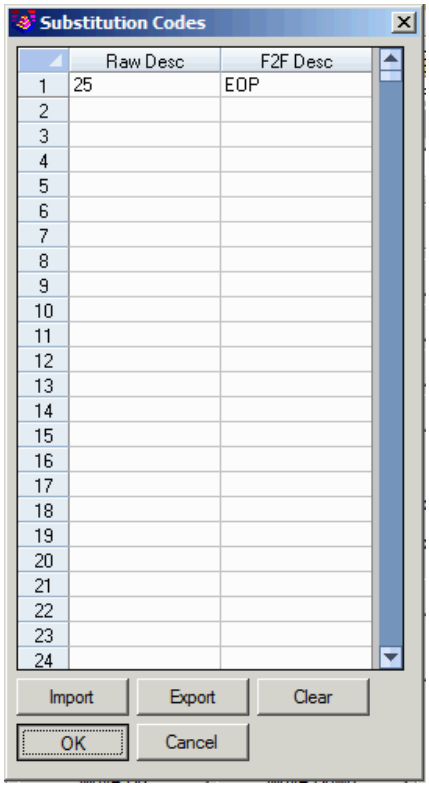
Max Delta-Height for Linework: Use this option to specify the maximum elevation difference that Draw Field to Finish should draw any section of linework. This option is for use with 3d polylines and lines.

Max Length for Linework: Specify the maximum length that Draw Field to Finish should draw any section of linework.

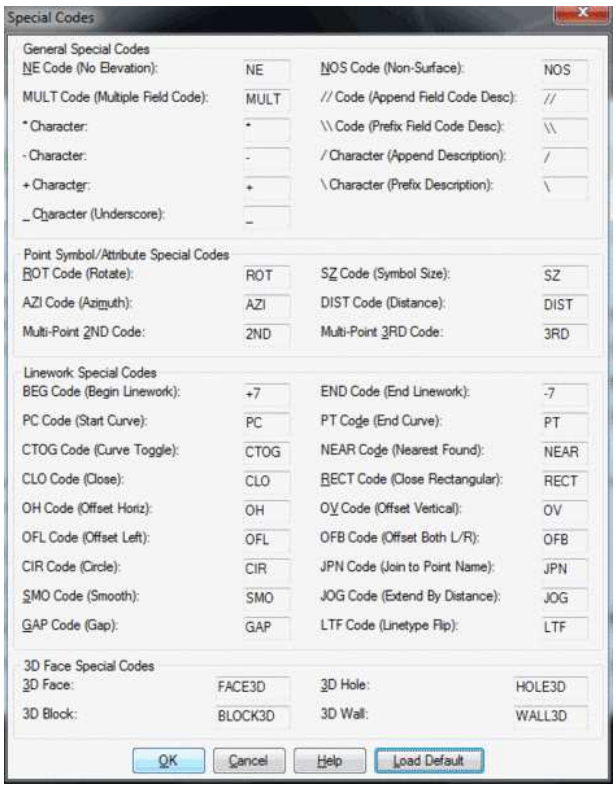
GIS Special Codes: This option allows you to use GIS attribute for Field-to-Finish special coding. For a select group of special codes, a GIS attribute can be assigned. When processing the points, if a point has GIS data for the specified attribute, then that attribute value is used for the special coding. For example, you can have a GIS attribute of COMMENT set to the Append Description special code. Then if a point has a GIS attribute for COMMENT, the value of that COMMENT will be added to the description label for that point.



Substitution Codes: This option defines a lookup table for translations of the raw point descriptions. This translation is done as a pre-processing step before the regular Field-to-Finish processing. For example, if you had a substitution setup for "25" = "EOP", then a point description of "25" would get translated to "EOP" and then this "EOP" would be processed with Field-to-Finish. Use the Import and Export functions to load and save substition codes to a comma separated text file.



Special Codes: This section allows you to substitute the existing predefined special codes and characters with your own. Draw Field to Finish recognizes several special codes. A special code is placed before or after the regular code with a space separating the code and special code. Here is a listing of the default special codes and characters.



Special Characters

The characters (∗, -, +, /, and _) can be used and substituted in Draw Field to Finish. The way these characters are used is that when the file is processed the description field is searched for these characters. If the "+" symbol

was changed to "-" then the program would look for "-" and change it to "+". This is useful when a particular data collector may not have all the symbols available. With these substitutions you can make a character that is provided on the data collector generate the symbol needed. Multiple characters can also be used. For example "–" can be used to in order to produce a "/" character or any of the characters listed above.

Special Codes

"/"

Carlson points in the drawing have point attributes including a description. When Field-to-Finish draws the points, the point description from the coordinate file is processed to match a code. The code then defines the description that is drawn with the point. For example, consider a code of "UP" with a description of "POLE" and a data point with the description "UP". The data point description "UP" would be matched with the code "UP" and the point would end up being drawn with the description "POLE". A special character "/" (the forward slash or divide key) can be used for an unprocessed description to append. Everything after the "/" is added directly to the point description and is not considered a code and no further substitution is done on it. For example, a data point with the description "UP / 150" with the same code "UP" definition above would be drawn with the description "POLE 150".

"\"
This special code takes the part of the description after the "\" and puts it as the prefix before the point description. For example, a data point with the description "TR \ 24ft" and a "TR" code definition with a description of "Tree" would be drawn with a description of "24ft Tree".

"//"

This special code causes text after the "//" to be interpreted as a field code. That field code's description is then appended to the first field code's description. For example, if the field code 02 has the description *2"* and the field code OAK has the description *oak tree*, then 02//OAK will result in the point having the description of *2" oak tree*. If the "/" character has been replaced with a different character, for example with a & character, then the "//" code would become "&&".

"\\"

This special code is the same as "//" except that field code's description is then prefixed instead of appended to the first field code's description.

MULT
This code applies when the Split Multiple Codes under Code Table Settings is set to None and you want to override this setting and explicitly spilt selected codes. Multiple codes apply to points with dual code definitions for drawing two different style points or for connecting different linework to the same point. For example, if a point is both a sidewalk and driveway corner, then the point description could be "SW MULTDR".

PC

This code begins a three point arc or a curved line when used with the "PT" code (see below). The point with this special code is the first point on the arc. The next point with the code is considered a point on the arc, and third point with the code is the arc endpoint. For example (in point number, X, Y, Z, description format),

10, 500, 500, 0, EP PC - start curve
11, 525, 527, 0, EP - second point on curve
12, 531, 533, 0, EP - end point of curve

PT

This is a special code that can be used with "PC" to define a curve with more than three points or a tangent two-point curve. Starting at the point with the "PC", the program will look for a "PT". If the "PT" is found, all the points between the "PC" and "PT" are used for the curve which is drawn as a smoothed polyline that passes through all points and only curves the polyline between points. If no "PT" is found, then the regular three point arc is applied as explained above. If no points are found between the "PC" and "PT", then the point prior to the "PC" and the point after the "PT" are used to create tangents for the resulting curve.

CTOG

This special code toggles curve mode on and off. Instead of using PC to start a curve, you can use CTOG. Likewise, instead of using PT to end a curve, you can use CTOG.

CLO

This code forces the lines drawn between a series of points with the same code to close back to the first point with the same code. For example, shots 1-4 all have the BLD description with the exception of point 4. Its description is BLD CLO. This will force the linework drawn for the BLD code to close back to point 1 which is the first point with the description of BLD.

GAP

This special code makes a single segment break in the current linework. For example, if you have a curb polyline that you want to break to skip over a driveway, then you could add the GAP code at the start of the driveway and continue the curb as normal on the other side.

NE

This code represents no elevation. A point with this special code is located at zero elevation.

NOS

This code indicates that the point should be "non-surface"; that is, that it should be ignored when contouring or creating surfaces. This can also be controlled per-field code by turning on the Non-Surface toggle in the Edit Field Code Definition dialog box.

Offsets: OH, OV, OFL, OFB

The codes "OH" and "OV" stand for offset horizontal and offset vertical. These offset codes apply to 2D and 3D polylines. A single set of offset codes can be used to offset the polyline a set amount. For example,

10, 500, 500, 100, EP OH2.5 OV-.5
11, 525, 527, 101, EP
12, 531, 533, 103, EP

This would create a polyline connecting points 10,11 and 12 and an offset polyline with a 2.5 horizontal and -0.5 vertical offset. The direction of the horizontal offset is determined by the direction of the polyline. A positive horizontal offset goes right from the polyline direction and a negative goes left. The horizontal and vertical offset amounts apply starting at the point with the offset codes until a new offset code or the end of the polyline. Only one horizontal and vertical offset can be applied to 2D polylines. For 3D polylines, multiple offset codes can be used to make a variable offset. For example,

10, 500, 500, 100, EP OH2.5 OV-.5
11, 525, 527, 101, EP OH5.5 OV-.75
12, 531, 533, 103, EP OH7.5

This would offset the first point horizontal 2.5 and vertical -0.5, the second point horizontal 5.5 and vertical -0.75 and the third point horizontal 7.5 and vertical -0.75.

When there are multiple "OH" codes for the same point, the polyline is offset multiple times.

The "OFL" code stands for offset left horizontal. The only difference with the "OH" code is that you don't have to enter the "-" to go left.

The "OFB" code stands for offset both left and right horizontal. For example, if the points follow the center of a ROW, the OFB code can be use to create the left and right edges of the ROW.

SZ

This code is used to set a different symbol size. There are several ways to use this code. It can take multiple scale factors for different dimensions by putting an ID character after the factor.

SZ: If nothing follows the SZ code, then the next point with the same field code as the current point will be used to determine the size.
SZ#: The value of the new symbol size is specified after the SZ. This value is the actual size in drawing units. For example, SZ2.
SZ#X: The value after the SZ is used to scale the symbol in the X dimension. For example, SZ2X.
SZ#Y: The value after the SZ is used to scale the symbol in the Y dimension. For example, SZ2Y.
SZ#Z or SZ#V: The value after the SZ is used to scale the symbol in the Z (Vertical) dimension. For example, SZ2Z.
SZ#H: The value after the SZ is used to scale the symbol in the X,Y (Horizontal) dimensions. For example, SZ2H.
SZ#S: The value after the SZ is a symbol size scaler that get multiplied by the drawing horizontal scale to determine the actual drawing units. For example, SZ0.2S.

The X, Y, Z, V and H can be combined. For example, to scale a symbol by 10 horizontally and 25 vertically, use SZ10H25Z. Or to scale a symbol by 2 in the X direction and 4 in the Y direction, use SZ2X4Y.

When multiple SZ codes are used in the same point description, the symbol is drawn multiple times at the different sizes. For example, a point description of "TREE SZ5 SZ10" will draw the tree symbol twice. One symbol will be size 5 and the other size 10.

ROT

This code is used to set the rotation of the point symbol. If a point number follows the ROT code, then angle from the current point to this point number is used for the rotation. For example, "ROT45" would rotate the symbol towards point number 45. If there is no point number after the ROT code, then the rotation point is the next point number with the same code as the current point or a companion code for the current code. ROT can also be used to rotate towards an angle clockwise from north by using ‘+' or ‘-' in front of the number. For example ROT+45 rotates the point symbol to the northeast and ROT-90 rotates the point symbol to the west.

SMO

This code is used to smooth the polyline.

AZI & DIST

The AZI and DIST codes are used together to locate an offset point. The AZI sets the offset azimuth and DIST sets

the distance. The values should directly follow the code. For example, AZI25 DIST4.2 would draw the point offset 4.2 at an azimuth of 25 degrees.

JOG

The "JOG" special code allows for additional points to be inserted into the line work at perpendicular or straight offsets. Only offsets should follow the JOG code. Positive numbers indicate a jog to the right and negative numbers indicate a jog to the left. Alternatively, "R#" and "L#" can be used where # is the distance to either the right or the left. Finally, "S#" can be used to make an offset straight ahead by using a positive # or behind by using a negative #. For example, "BLDG JOG S10.1 R5 L12.2 L5 L12.2" or equivalently "BLDG JOG S10.1 5 -12.2 -5 -12.2" advances 10.1 units and then draws a closed rectangle on the right hand side of an existing line. The offsets are always done in the X-Y plane. If the current line is vertical, an offset to the right is along the positive X-axis.

JPN

The "JPN" (Join to Point Name) special code joins to the point named immediately after the code. For example, "JPN205" causes a line to be drawn from the current point to the point "205". JPN is designed to work for adding a segment at the start of linework. So the point with the JPN code should be at first segment of the linework.

NEAR

This special code sets the current polyline to Nearest Found connection order. This applies to codes that have the Connection Order set to Sequential and you want to override this setting to Nearest Found for the current polyline.

RECT

The "RECT" special code causes a rectangle to be formed on a 2D or 3D polyline using one of two different methods. If a number follows "RECT" (e.g., "RECT10"), a rectangle will be drawn 10 units to the right of the last two points ending on the point with the "RECT" code. Use a negative offset to place the rectangle on the left side (e.g., "RECT-2.5"). For example if locating the left side of a 10' rectangular concrete pad using the code conc for concrete, the description of the two left points would be (conc) for the first point and (conc rect10) for the second. If no number follows "RECT", then the polyline will be closed by shooting right angles from the first point of the polyline and the current point and creating a new point where those two lines cross. This method requires three points be established on the pad.

LTF

The "LTF" (LineType Flip) special code switches the side for the linetype. This option applies to non-symmetrical linetypes like the treeline or guard rail for when you want the linetype to face the other way.

CIR

The "CIR" special code stops the linework on the previous point and causes this point to create a circle in one of three different ways. The first way uses just the current point as the center with the CIR special code followed immediately by the radius. For example "CIR5.0" will create a circle centered on this point with radius 5 and at the elevation of the current point. The second method uses two points, the first point specifying the center and the elevation, and the second point specifying the radius. Only the first point has the "CIR" code. The third method uses 3 or more points that specify the perimeter of the circle in 2D with the first point specifying the elevation. For this method, the "CIR" special code is only on the first point.

The "CIR" code can be used with all of the linetypes including "points only". The circles are always parallel to the X-Y plane.

For Multi-Point 2ND Code

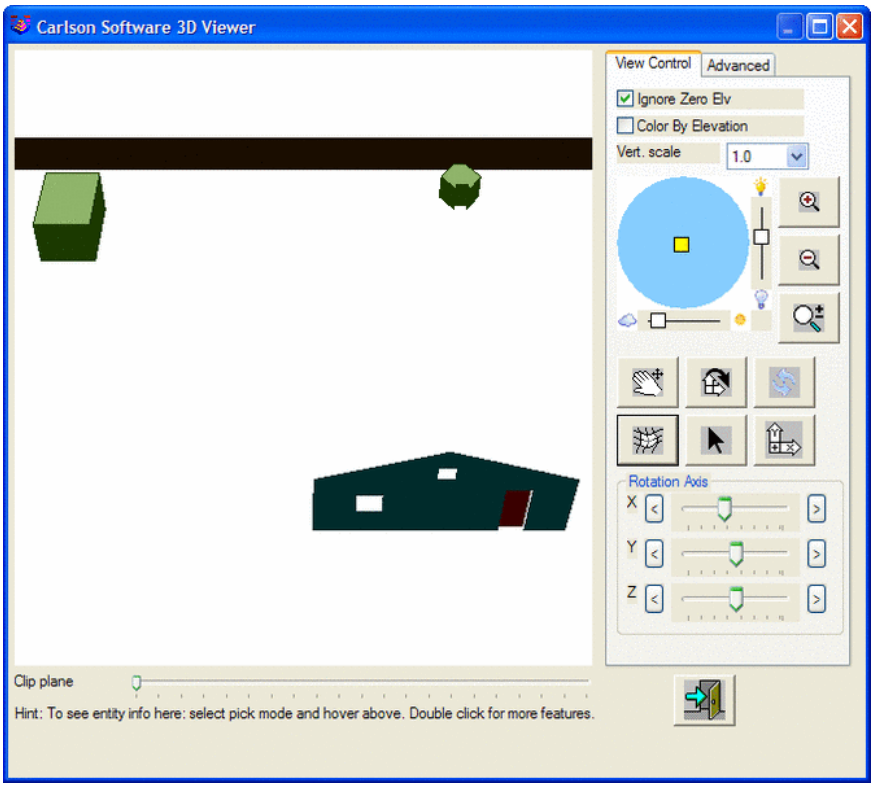
When used on the first point of a multi-point symbol, the "2ND" code indicates that the second point of the sequence (i.e., the next point after the current one) should be used as the second symbol insertion point for a multi-point symbol. Please refer to Symbol Pts in the Edit Field Code Definition section below.

For Multi-Point 3RD Code

When used on the first point of a multi-point symbol, the "3RD" code indicates that the third point of the sequence should be used as the third symbol insertion point. The "3RD" code should be used with the "2ND" code. Please refer to Symbol Pts in the Edit Field Code Definition section below.

3D Special Codes

Below are the special codes that can be used for the easy creation of 3D surfaces. The resulting 3D face entities can be viewed in the Carlson 3D viewer by entering "cube" on the command line.



FACE3D

Makes a triangle mesh of 3D face entities by triangulating points starting with the current point and continuing until the line ends or another 3D special code is found. The points must be ordered along the perimeter. Although the mesh will be built if the points are clockwise or counterclockwise along the perimeter, the visible side in the Carlson 3D viewer, "cube", is the clockwise side by default. On the Advanced tab, the shading mode may be set to *Shade both* or *Shade back* if you would prefer to see both sides or just the counter-clockwise side.

HOLE3D

Makes an exclusion area within the triangle mesh identified by the point number following this code (e.g., "HOLE3D101" will start a hole in point # 101). If no point number is given ("HOLE3D"), the exclusion area is applied to the last mesh or if there is a mesh in the process of being constructed by the current sequence of points, it is ended and the hole is applied to it. Note that a hole can only be applied to a mesh that was created by FACE3D (not BLOCK3D or WALL3D). Note also that it can be difficult to predict what the "last mesh" was if it used a different field code since the points of the coordinate file are processed by order of field code first and then point number. There is no limit to how many holes can be applied to a FACE3D mesh. The points of the hole itself are

not added to the FACE3D mesh; they are projected on to the best plane that contains the FACE3D mesh and then the hole is cut-out.

Example 1:
2500 HOUSE1 FACE3D */front of house*
2501 HOUSE1
2502 HOUSE1
2503 HOUSE1
2504 HOUSE1
2505 VENT1 HOLE3D2500 */applies 2505-2508 as a hole to last mesh that uses point #2500. So any point in the range 2500-2504 would have the same effect.*
2506 VENT1
2507 VENT1
2508 VENT1

Example 2:
2500 HOUSE1 FACE3D */front of house*
2501 HOUSE1
2502 HOUSE1
2503 HOUSE1
2504 HOUSE1
2505 HOUSE1 HOLE3D */stops the above mesh and applies 2505-2508 as a hole*
2506 HOUSE1
2507 HOUSE1
2508 HOUSE1

Example 3:
2500 HOUSE1 FACE3D */front of house*
2501 HOUSE1
2502 HOUSE1
2503 HOUSE1
2504 HOUSE1
2505 WINDOW1 FACE3D HOLE3D2503 */applies 2505-2508 as a hole to above mesh 2500-2504 and starts a new mesh using the WINDOW field code.*
2506 WINDOW1
2507 WINDOW1
2508 WINDOW1

Example 4 (same result as Example 3):
2500 HOUSE1 FACE3D */front of house*
2501 HOUSE1
2502 HOUSE1
2503 HOUSE1
2504 HOUSE1
2505 WINDOW1 FACE3D */starts a new mesh using the WINDOW field code.*
2506 WINDOW1
2507 WINDOW1
2508 WINDOW1 HOLE3D2504 */makes the mesh 2505-2508 also be a hole in the mesh 2500-2504.*

BLOCK3D

Makes a set of 3D faces to make a 3d block using the height value entered after the code (e.g., "BLOCK3D2.3" with height 2.3). Heights can be positive or negative. With 3 points, makes a parallelogram base that is extruded up (or down if height is negative) to form a 6-sided block, including top and bottom. With 4 or more points, makes a closed

polygon for the base that is then extruded by the height. The points can be laid out in clockwise or counterclockwise order around the perimeter. The perimeter or base does not have to be a convex polygon.

WALL3D

Makes a set of 3D faces above the polyline using a height value entered after the code (e.g., "WALL3D2.3" with height 2.3). The height can be negative if the points on the top of the wall have been shot. If no parameter exists, then the height is determined by the distance from the current point to the next point. This is a signed distance so the surveyor can shoot either the top of the wall or the bottom of the wall. Both sides of the wall will have triangles and so both sides will always be visible in the Carlson 3D viewer "cube".

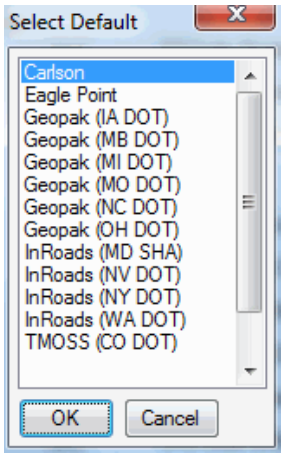
Example 5 – 6' high wall shot along the bottom:

2000 1000.000 1060.000 100.000 WALL1 WALL3D6.0 */wall 6'*
2001 1100.000 1060.000 100.000 WALL1
2002 1100.000 1160.000 100.000 WALL1

Example 6 – 6' high wall, height specified by 1st to 2nd point, shot along the top:

2020 1100.000 1160.000 100.000 WALL2 WALL3D */height by 2nd pt*
2021 1100.000 1160.000 106.000 WALL2
2022 1000.000 1160.000 106.000 WALL2

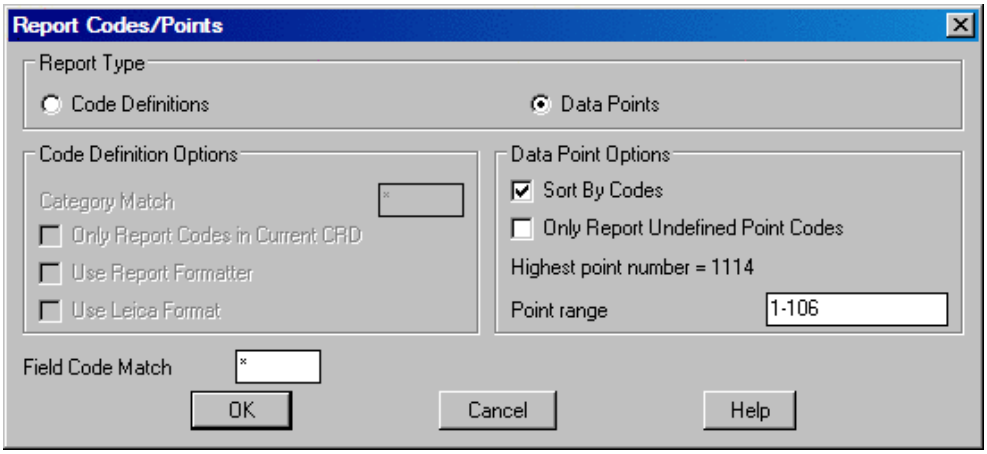
Load Default
This button sets the special codes to Carlson, Eagle Point, Geopak, InRoads or TMOSS defaults.



Code Table (continued)

Sort Table - This sorts the code table by either code name or layer.

Report Codes/Points - This routine prints the code table or the data file to the screen, file, or printer. A useful option here is to print the data file (CRD Points) and choose Sort by Codes which will group the data points by distinct codes.



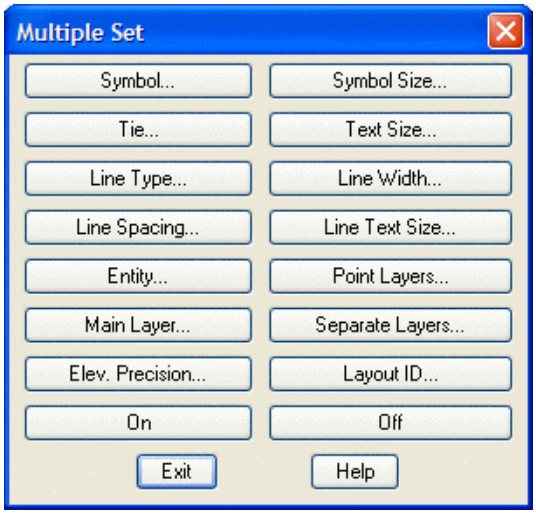
Code Table by CRD - This command will create code table definitions based on the coordinate file field descriptions. This is useful when creating a code table from scratch.

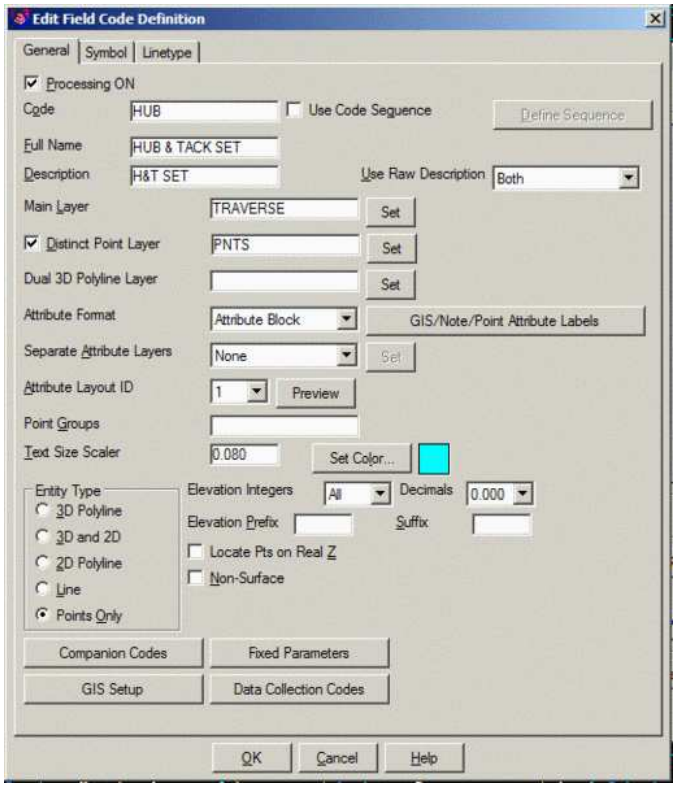
Save: Saves the Draw Field to Finish field code definition (.FLD) file.

Save As: Reacts the same as Save but allows for specification of file name and location to save to.

Code Definitions

Edit: If only one field code is selected, then this command opens the Edit Field Code Definition dialog box. If multiple field codes are selected (by holding down the control key or shift key and clicking on the rows), then the Multiple Set dialog box will open.





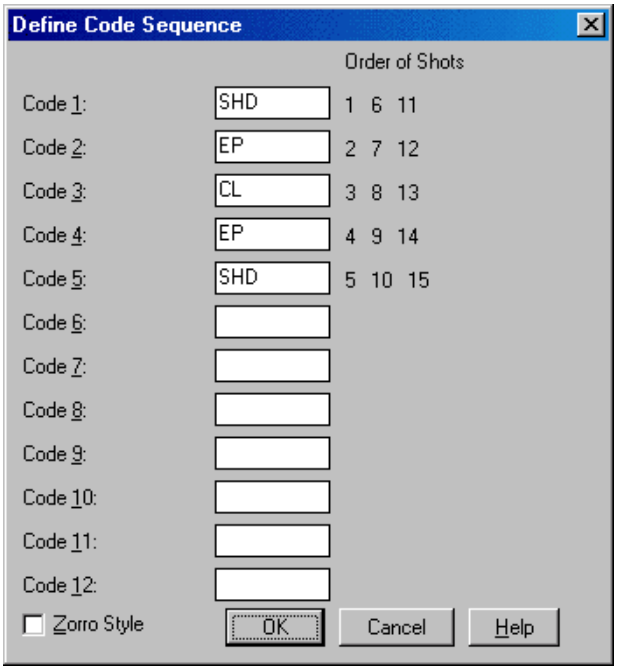
The code definition dialog has three tabs: General, Symbol and Linetype. Here are the settings under General:

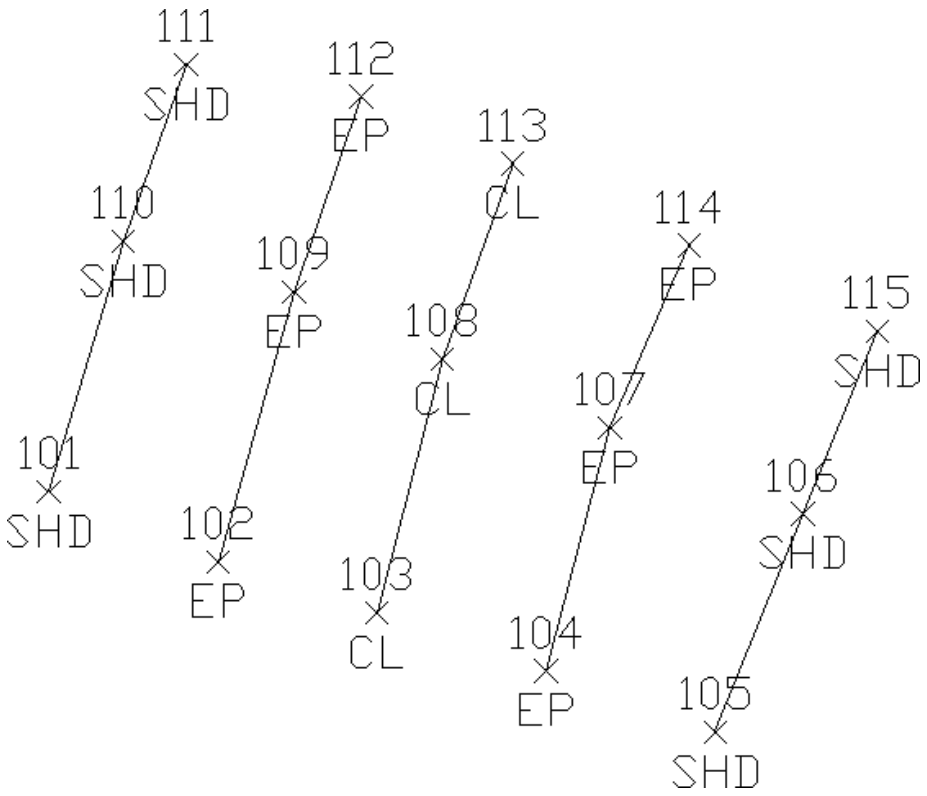
Processing ON: This toggle controls whether this code will be processed.

Code: This is the key name that identities the code and is matched with the field data descriptions. It is important to note that the $*$ character, used in this field, is regarded as a wildcard or "match anything" code. For example, a field code definition with the code defined as TREE$*$ will be used for any raw description of TREE. Raw descriptions of TREEA, TREE12, TREE, etc. will match the TREE code definition. This will always be the case unless there is a more specific code is found. For example is there was a code TREEA in the code definition file, then that code would be used instead of the TREE code.

Use Code Sequence: This specifies a sequence type code. Sequences are a way to simplify field entry of a sequence of codes. For example, a road cross-section could be SHD1 EP1 CL EP2 SHD2. Instead of entering these different descriptions, one sequence definition can store these descriptions in order. Then just the sequence code (such as RD) is used in the field. The cross-section can be shot in left to right then left right order, right to left then right to left order, or alternating left to right then right to left order. The alternating method is known as the Zorro style. The one restriction is that the shots always start from a right or left edge.

To set up a sequence, choose the Sequence toggle in the Edit Code dialog. Then pick the Define Code Sequence button. This brings up a dialog for entering the sequence codes in order. These sequence codes should be defined as normal codes somewhere else in the Draw Field to Finish code table (ie SHD as a 3D polyline). In the field, the one template code is used for all the cross-sections shots (ie RD for all the points). Then Draw Field to Finish will substitute this template code with the sequence codes (ie substitute RD with SHD).





Resulting points and linework showing Zorro style template **Define Code Sequence:** This sets the code names that make up the sequence.

Full Name: This is an optional field that describes the code for viewing.

Description: This value is assigned to the point description attribute when the point is drawn. This description can be different than the field description. An additional description can be added to a point by entering it after a forward slash in the data description field.

Use Raw Description: This option turns off the Description field described above. Instead the points will be drawn with their original unprocessed descriptions. The Attribute Block option applies to the point block with

point #, elevation and description fields. The Text Attribute applies to drawing the description as text. The format of the description is controlled by the Attribute Format setting.

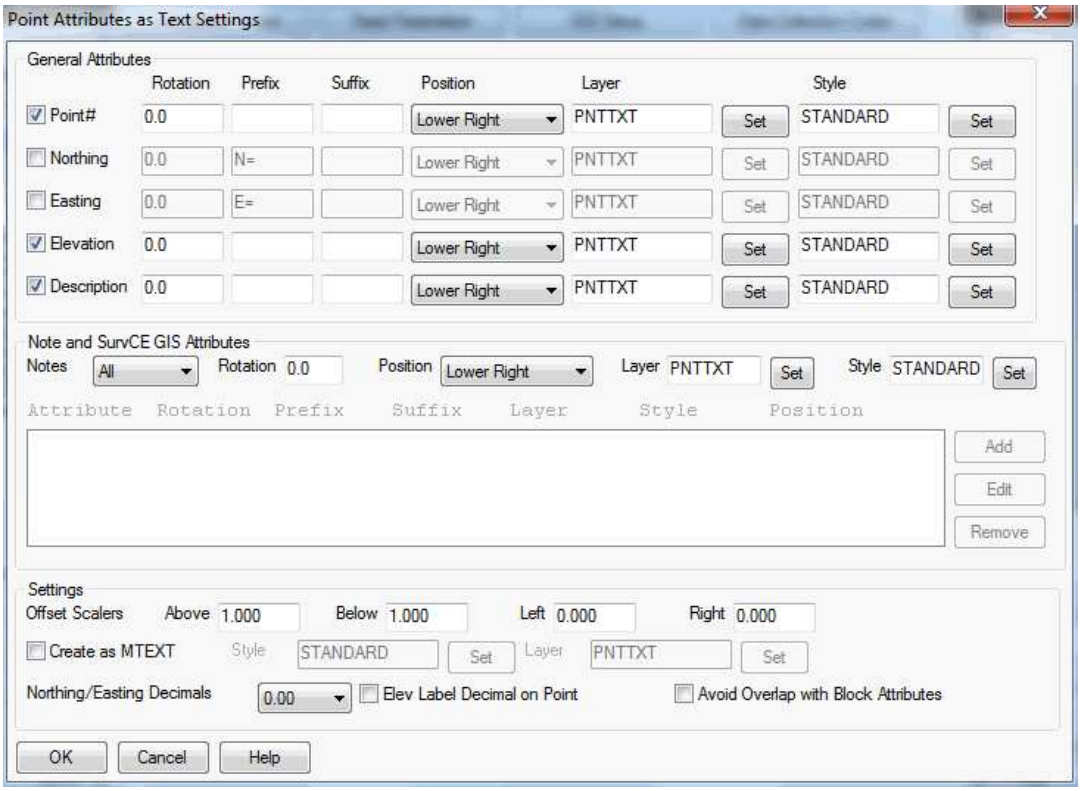
Main Layer: The point and line work for the code will be created in this layer.

Distinct Point Layer: When this toggle is selected, the line work is created in the layer defined in the Layer field and the points are created in the specified distinct point layer. For example, you could have DRIVEWAY for linework and DRIVEWAY_PNT for the points.

Dual 3D Polyline Layer: Displays the layer that the 3d polyline will drawn on when using an Entity Type of 3D and 2D. The layer name can be typed in this field.

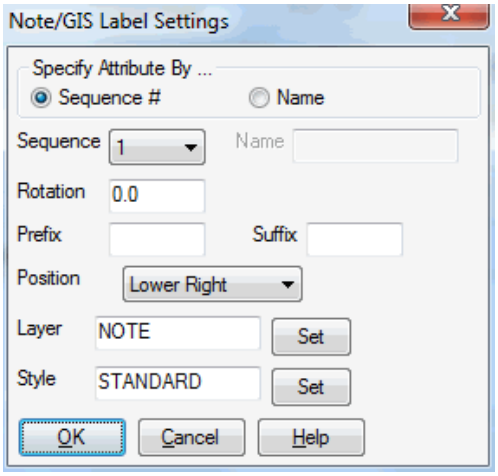
Set 3D Layer: Sets the layer that the 3d polyline will drawn on when using an Entity Type of 3D and 2D. The layer can be selected from the list or typed in at the bottom of the dialog box.

Attribute Format: This chooses the type of point entities to create. The Attribute Block format creates the Carlson point entity which is block with attributes for point#, elevation and description. The Text Attribute format creates text entities for each of the point attributes. When the Text Attribute format is selected, the Set button is available where you can control which attributes to draw as text and the position, rotation, decimals, style, prefix, suffix and layer for each attribute. The Offset Scalers control the distance for the text from the point for the different positions. These offset distances are calculated by multiplying the scaler by the horizontal scale for the drawing. The Avoid Overlap With Block Attributes option expands the offset distance starting point from the point to the bounding box that encloses the point block attributes.



Also, for points notes and SurvCE GIS attributes, you can choose to all or selected fields. For selected, use the Add, Edit and Remove buttons to build the list of fields to label. To specify the field to label, the Sequence# method sets the field by its order position. For example, a sequence of 3 would use the third attribute for the point. The Name method sets the field to label by field name such as HRMS.

For each field, there are settings for the rotation, prefix, suffix, position, decimals, layer and style. The decimals setting applies to GIS fields that are real numbers.



Besides labeling attributes as text with this method, the Custom Attributes feature is a way to label attributes as block attributes.

Separate Attribute Layers: This controls the layers of the point and symbol attributes. With "None" the point layers are the standard layers, "PNTNO", "PNTELEV" and "PNTDESC", and the symbol layer is "PNTMARK". With "Points" or "Both" the point attribute layers begin with the layer for the code followed by the attribute type. For example, the "DWL" code shown in this dialog has a layer name "DRIVEWAY". The point attributes would then be "DRIVEWAYNO", "DRIVEWAYELEV" and "DRIVEWAYDESC". With "Symbols" or "Both" the symbol attribute layer begins with the layer for the code followed by "MARK".

Attribute Layout ID: Controls the location of the point number, elevation and description. These attribute layouts are defined in the drawings that are stored in the Carlson SUP directory with the file name of SRVPNO plus the ID number (i.e. SRVPNO1.DWG, SRVPNO2.DWG, etc.). If you want to change the attribute positions for a layout ID, then open and edit the associated SRVPNO drawing.

Point Groups: This field is for the name of the point group that all points with this code will be added to. If the points for this code belong to multiple point groups, you can specify multiple point group names in this field separated by commas. Under Draw in Additional Draw Options, there is an option whether to automatically use the code name as the point group name or to use the name defined in the code definition.
Text Size Scaler: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size.

Set Color: The line work will be drawn in this color. The default is BYLAYER.

Entity Type: This defines the line entity to be created. Points only does not create any line work. 3D Polyline can be used for breaklines. 3D and 2D entity type selection creates a 3d polyline in the layer specified in the Dual 3d polyline layer setting and a 2d polyline in the layer identified in the Layer setting. Since 3d polylines do not display linetypes, this is useful when needing linework in 3d for design work while also needing to display linetypes for final plotting of the drawing. This provides an easy and quick way to turn off all 2d polylines or all 3d polylines by using the layer control dialog or the appropriate toggles in the Draw Points dialog.

Elevation Integers: This controls the number of digits to display to the left of the decimal point for the elevation label. The All setting will show the full elevation digits. The other settings allow you to limit the number of digits to display for the purpose of reducing the amount of space the elevation labels take up in the drawing. For example, if a site is in the 4000 foot elevation range, then this setting could be set to three digits (000) and an elevation of 4321 would be labeled as 321.

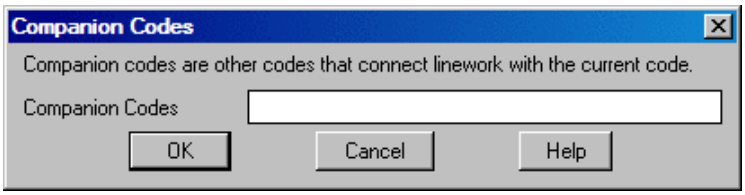
Elevation Decimals: This controls the display precision for the elevation label.

Elevation Prefix/Suffix: These set the prefix and suffix for the elevation label per code. In the Draw function under Additional Draw Settings, there is an override to set the elevation prefix/suffix for all the codes.

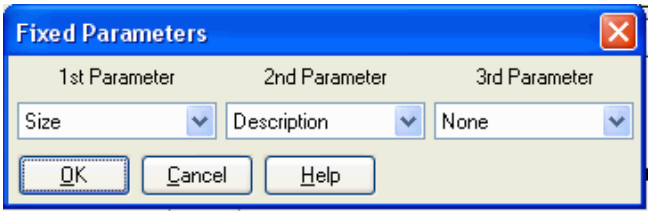
Locate Pts on Real Z Axis: This option will draw the points at the actual point elevation. Otherwise the points are drawn at zero elevation. For example, you could turn this option off for the FH for fire hydrant code to drawn them at zero. Then the GND code could have this option on to draw the ground shots at their elevations.

Non-Surface: Entities created with this flag are ignored when contouring or creating surfaces regardless of their elevation.

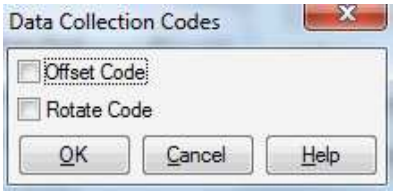
Companion Codes: This option allows different codes to connect when defined as line, polyline or 3d polyline. For example, a main line power pole code may be defined as PP while a service utility pole may be defined as UP. When processing Draw Field to Finish, it may be desired to connect all PP and UP codes together. This could be accomplished by defining a companion for UP as PP and a companion code for PP as UP. Each code needs to reference the other as a companion code.

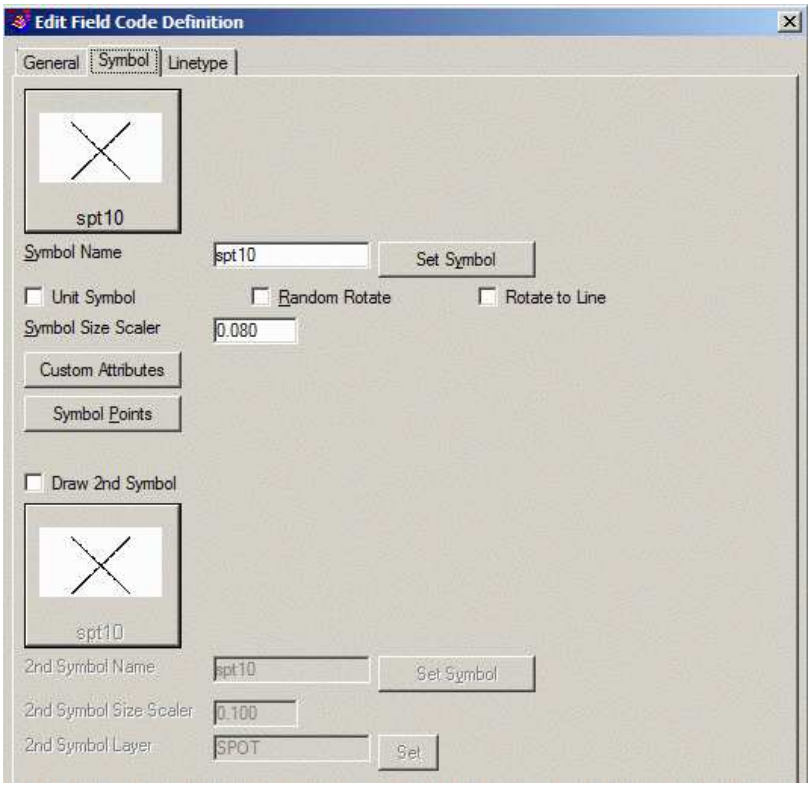


Fixed Parameters: This option is a coding method where you specify a sequence of parameters that follow the main code. There can be up to three parameters and these parameters can be an additional description or special codes Size, Rotate, Azimuth, Distance or Offsets. The purpose for Fixed Parameters is to save keystrokes by not having to enter the special code prefix. For example, for a code TR for Tree along with a size 12 feet and description of Oak, the special code description would be "TR SZ12 // OAK". With Fixed Parameters of Size and Description, the description would be "TR 12 OAK".



Data Collection Codes: These settings apply to Carlson Field for turning on the Offset mode and Rotate mode automatically by F2F code.





Here are the settings on the Symbol tab:

Set Symbol: This is the point symbol for the code. The dialog allows you to select from the symbols defined in the Symbol Library which is setup with the Settings->Symbol Library command. Besides the symbols from the symbol library, you can also use any symbols that are defined as blocks in the current drawing by entering the block name in the symbol edit box. To have a point without a symbol, use the Carlson symbol named SPT0 which represents "no symbol". **Unit Symbol:** This option will draw the point symbol at unit (1:1) scale. For example, this option could be used for a symbol that is already drawn to actual dimensions such as a car symbol.

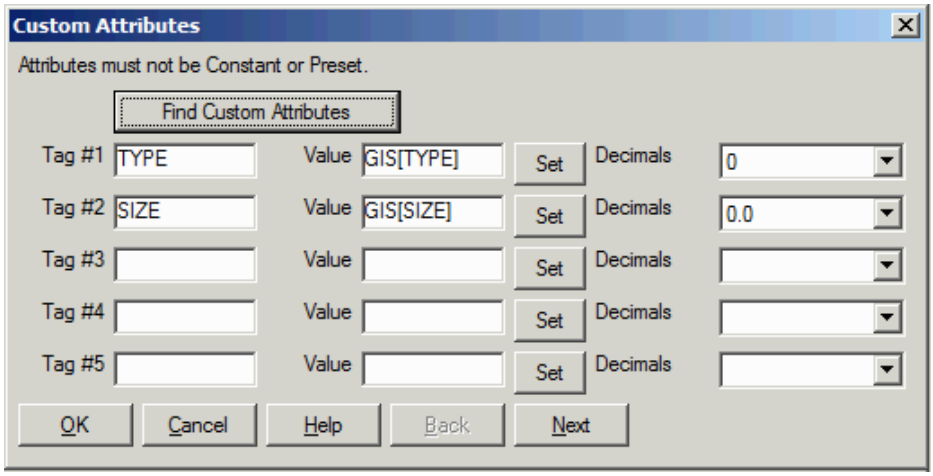
Random Rotate: This option will randomly rotate the symbol. For example, this option could be used for tree symbols to have the trees drawn in various orientations.

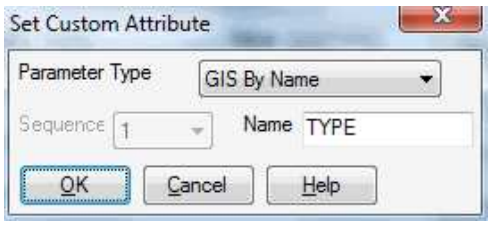
Rotate To Line: This option applies to points that are part of Field-to-Finish linework. This option will align the point attributes and symbol to the associated linework.

Symbol Size Scaler: This is a scaler value that is multiplied by the horizontal scale to obtain the actual size in the drawing. The horizontal scale can be set in *Drawing Setup*.

Custom Attributes: This feature allows you to use customized blocks that have customized attributes (the tag/value pairs). This feature works for both point attribute blocks and symbols. For attribute blocks, Field-to-Finish looks for attributes with the tags "PT#", "ELEV2", and "DESC2". The custom attributes feature allows you to define additional attributes in their custom blocks on a per-field code basis. The dialog shows five attributes at a time. The number of attributes is unlimited. Use the Next and Back buttons to show more attributes.

For an example, the custom block could have an attribute with the tag "TREE_SPECIES" and there is a separate field code for each species of tree. Each of those field codes can specify the value that should be assigned to the attribute that has the TREE_SPECIES tag. Then when the points are drawn, the tree species is shown. Note that the custom attributes must have their Constant and Preset properties set to "no". The custom attributes settings in F2F should not use those tags that the software already handles (PT#, ELEV2, and DESC2), or the setting will be ignored.

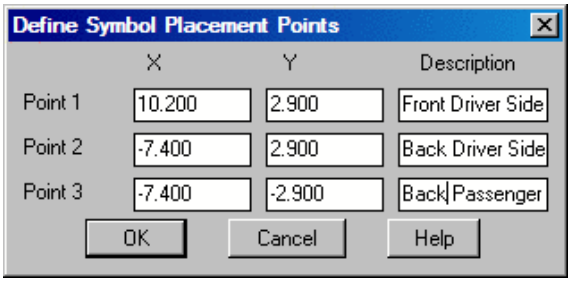


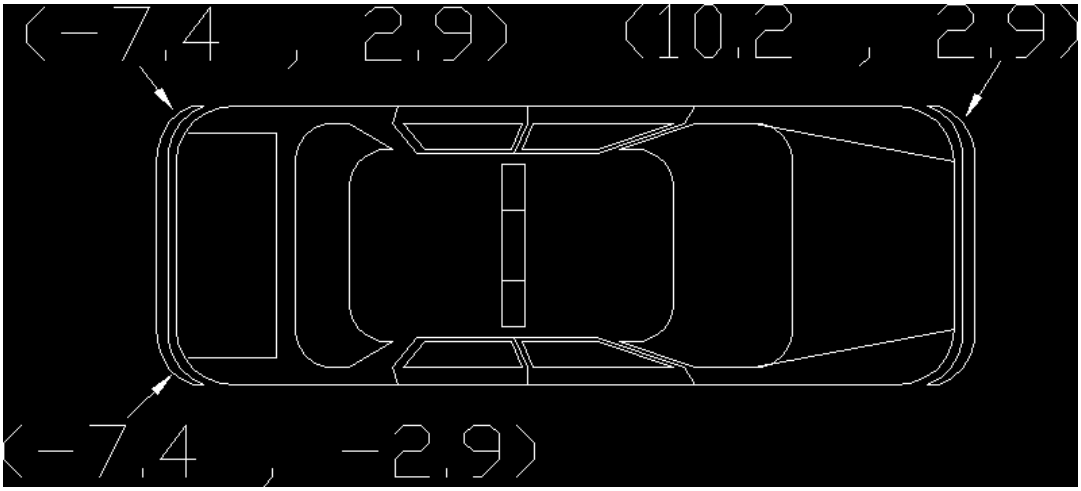


The Values for the attributes can be fixed strings that you enter in the dialog shown here. Or they can be dynamic parameters including point#, northing, easting, elevation or description for the current point as well as a point note or GIS attribute. To setup a parameter value, pick the Set button and then select the attribute. The Decimals setting applies to fields that are real values.

Besides labeling as block attributes, the Attribute Format method of Text mode is a way to label the attributes as text entities.

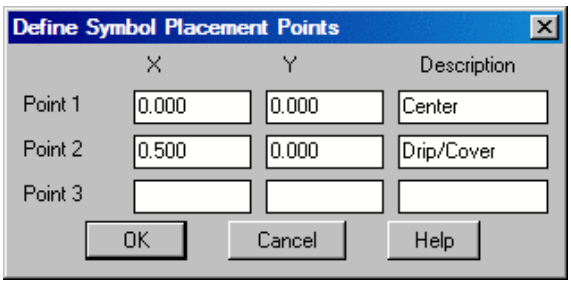
Symbol Points: For each code definition, the symbol insertion points can be defined with up to three points. To define the symbol insertion points, choose the Symbol Pts button in the Edit Code Definition dialog box. By default, the symbol insertion is defined by one point at the symbol center (0,0). A one point insertion definition can be used to insert a symbol offset from the center. With a two insertion point definitions, the program will rotate and scale the symbol. For example, two insertion points can be used to insert a tree symbol to size the tree, where the first point is for the tree center and the second is for the drip line. With three insertion point definitions, the program will rotate and scale the symbol in both X and Y. For example, three points can be used to insert a car symbol with the first point being the front drivers side, the second point as the back driver side (to rotate and scale the length) and the third as the back passenger side (to scale the width). Besides the insertion point coordinates, you can define a description for each point which is used for the drawn point description and is used for prompting in the Insert Multi-Point Symbol command and in Carlson Field data collection.

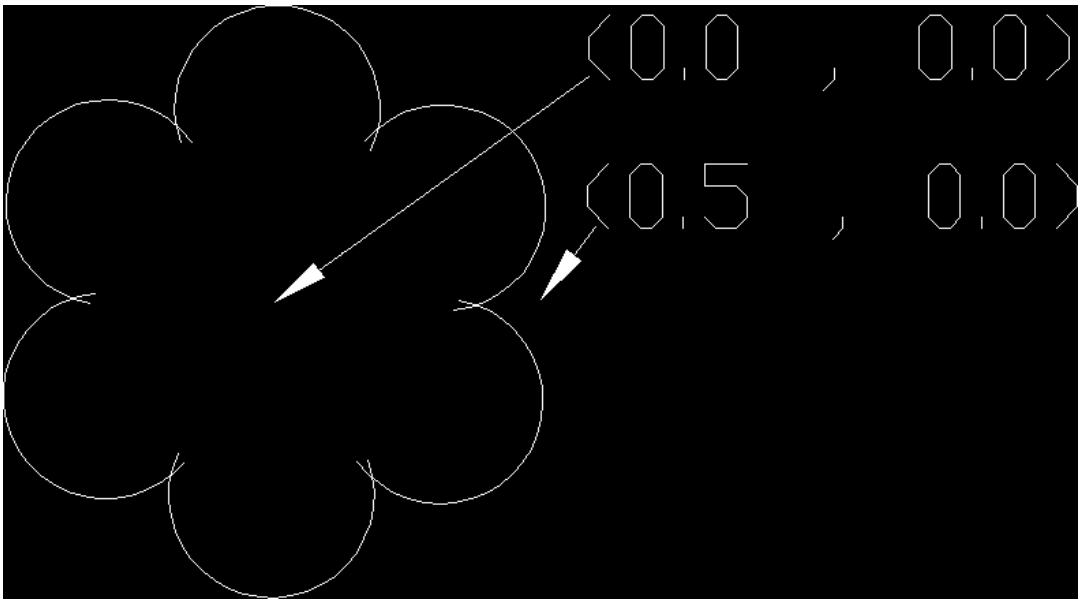




Three Point Symbol Drawing

The coordinates for the insertion point definitions are for the symbol at unit size. To figure these coordinates, you will need to open the symbol drawing (.DWG) file. By default, the symbols are located in the Carlson SUP directory. For example to make an insertion point for the tree drip line, open the tree symbol drawing and find the coordinate at the edge of the tree symbol (in this case 0.5,0.0).

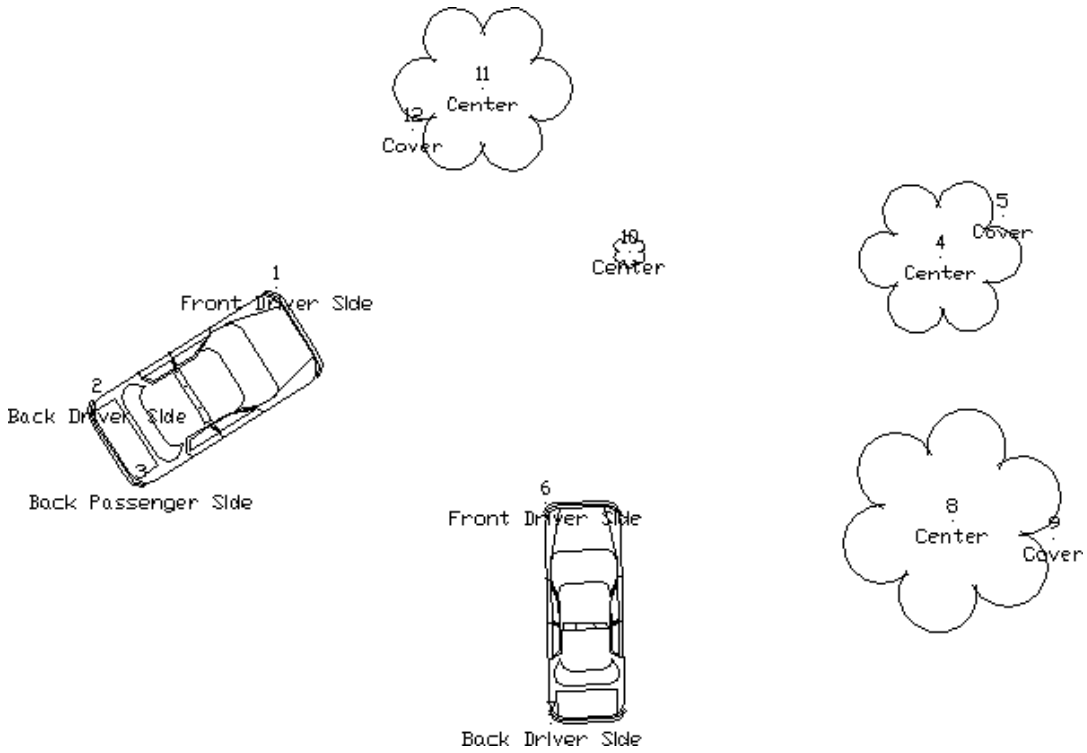




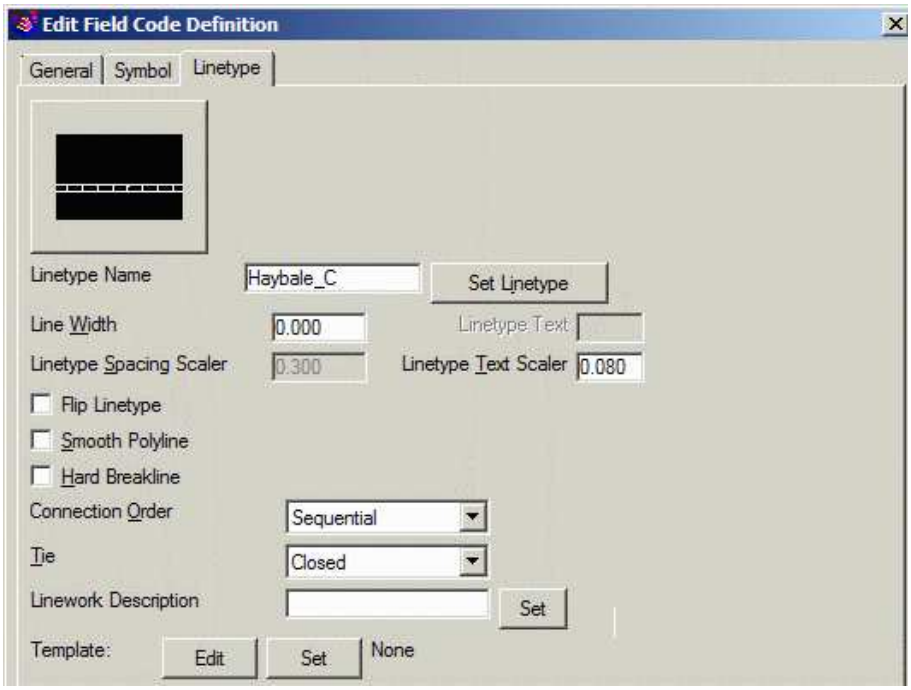
Two Point Symbol Drawing

Not all of the symbol insertion points need to be used when drawing the points. If a code definition has a three insertion points, it is possible to use just the first two or first one. There are special codes to associate multiple points to the same symbol. The first code point is used as the first symbol insertion point. The "2ND" code is used to specify the second symbol insertion point. A point number can follow the "2ND" to identify a specific point. Otherwise without the point number, the program will use the next point with the current code. The "3RD" code is

used to specify the third symbol insertion point and similar to the "2ND" code, a point number after the "3RD" is optional. The "2ND" and "3RD" codes should be assigned to the first point. For example, consider a code of "CAR" with a three point symbol insertion definition. If point #1 has a description of "CAR 2ND 3RD", then point #1 will be used as the first symbol insertion point and the next two points with the "CAR" description will be used as the second and third symbol insertion points.



Multi Point Symbol Drawing**Draw 2nd Symbol:** This option creates a second symbol on each point. This additional symbol can be used to add a 3D symbol to a 2D symbol used as the first symbol. Besides selecting the symbol name, there are settings for the symbol size and layer.



Here are the settings on the Linetype tab:

Set Linetype: Line work can be drawn in any of the special linetypes or with the linetype for the layer ("BYLAYER"). There are three types of pre-defined linetypes: CAD, Entity and Continuous. The type is shown as part of the linetype names in the list. The CAD linetypes are the default linetypes available in AutoCAD and IntelliCAD. The Entity linetypes insert text or symbol entities at the linetype interval. These linetypes are the same as used with the Annotate->Polyline To Special Line command. The Continuous linetypes define a special linetype in CAD and create continuous polylines with that special linetype. These linetypes are the same as with the Annotate->Change Polyline Linetype command. Besides these pre-defined linetypes within Field-to-Finish, you can also use any linetype that is defined in the drawing by entering that linetype name in the linetype edit box or by picking the Select From Drawing button within the Set Linetype dialog. The spacing and size of the special linetypes is determined by the CAD LTSCALE system variable and by the field code settings *Line Type Spacing Scaler* and *Line Type Text Scaler*. The special linetype "hedge" is drawn with a user specified width. You will be prompted for this information when you select that linetype. The special linetype "userdash" is drawn with user specified distances for the length of the dash and the length of the gap between dashes.

Line Width: This controls the width for the linework. Only applies to 2D polylines.

Linetype Text: This is the text that is used for the user-defined linetype. First use Set Linetype to either Other_E or UserDef_C. Then this text will be used for the linetype. For example, if you have a code for a 8" PVC pipeline, then you could set this text to 8" PVC.

Linetype Spacing Scaler: This is a scaler value that is multiplied by the CAD LTSCALE system variable to give the distance between symbols in the line.

Linetype Text Scaler: This is a scaler value that is multiplied by the CAD LTSCALE system variable to give the size of the text in a line.

Flip Linetype: This option switches the side for the linetype which applies to non-symmetrical linetypes like the treeline or guard rail.

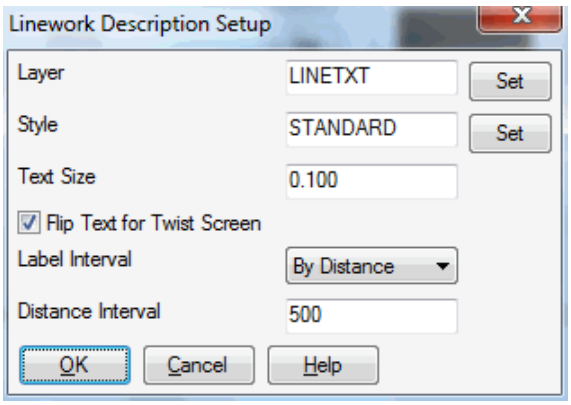
Smooth Polyline: This applies a modified Bezier smoothing to the polyline. The smoothed polyline will pass through all the original points.

Hard Breakline: This will tag the 3D polylines created with this code as hard breaklines. In *Triangulate & Contour*, contours are not smoothed as they cross hard barriers.

Connection Order: The points of a distinct code can be connected in their point number order or by nearest found which makes the line by adding the next closest point.

Tie: When checked the linework drawn with this code will always close. For example if you have points 1, 2, 3, and 4 with the code BLDG and Tie is checked on for the code BLDG, then the linework will be drawn from point 1 to 2 to 3 to 4 and then back to point 1, closing the figure.

Linework Description: This description is labeled along linework created by this code. The Set button displays a dialog to control the layer, style and size for these labels. You can also set the label interval.



Set Template: For 3D polyline codes, this option allows you to assign a template (.TPL) file to the code. The code points act as the centerline for the template and the program will drawn parallel 3D polylines for each break point (grade ID) in the template. The template file is defined in the Civil Design module.

Select All: This option selects all the codes. This can be used when only wanting to process a couple of codes. For example, use the select all option to select all the codes and then turn them off. Now select the codes for processing and turn them on. Also it can be used to make a global change to all the codes.

Add: The new code definition is inserted in the list in the position after the currently selected one. If none are selected for positioning, the new code is placed at the top. Only one code definition may be highlighted before running this routine. **Copy:** This option copies the definition of a selected code. It opens the Edit Field Code Definition dialog and copies the definition of the selected code to the appropriate settings. It does not copy the name of the code. It is a time saving tool to use when creating codes that are similar with only a couple of differences. **Cut:** This command will remove the highlighted code definitions from the list and puts them in a buffer for retrieval with Paste. **Paste:** This command will insert the code definitions put in the buffer by the Cut command. These codes will be inserted after the row of the currently highlighted code or at the top. **Search:** Allows you to search for a specific code in the list.

Coordinate File

Set CRD File: This command allows you to specify a coordinate (.CRD,.CGC,.MDB,.ZAK) file to process.

Edit Points: This command opens the *Edit Points* spreadsheet editor. See *Edit Points* for more details.

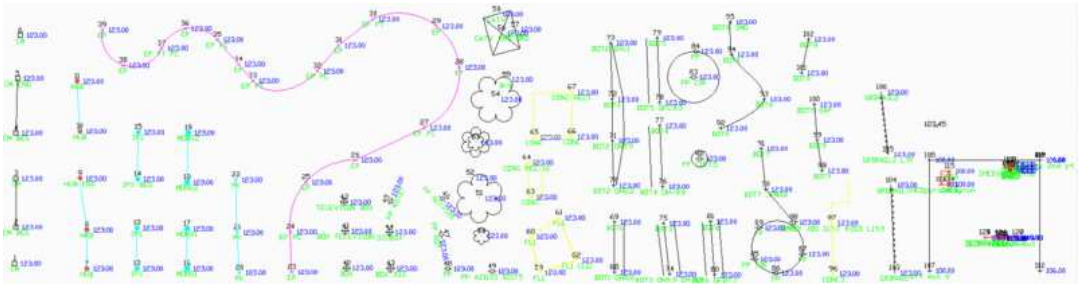
Draw: This command returns to the Draw Field to Finish dialog box.

Coding Examples

Under the Carlson Projects folder, there is an example that shows the different ways for linework coding along with examples for many of the special codes. The examples are in f2f_example.crd and f2f_example.fld. Here is a breakdown of the features that the points illustrate.

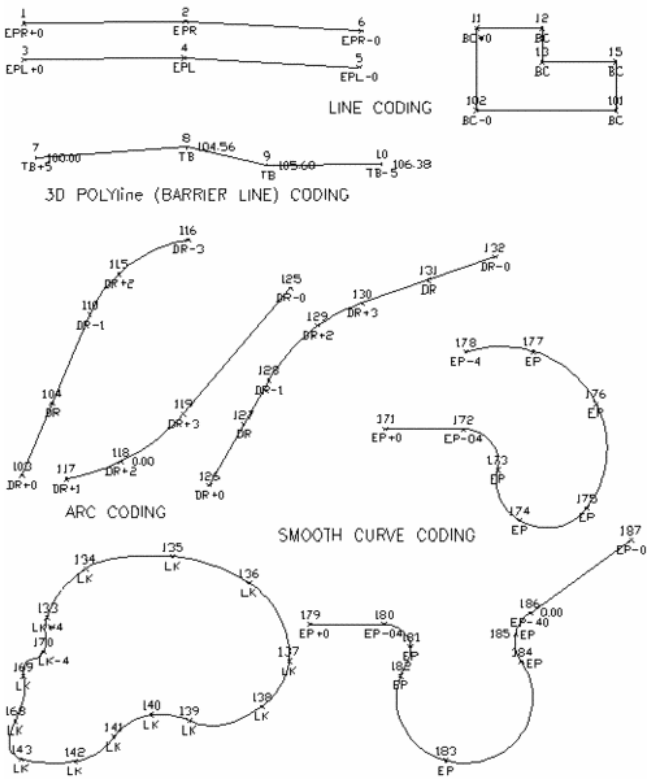
Point 1: Point Entity by itself
Points 2-3: Using Begin code to start a line; end line using Begin code for next line
Points 4-5: Using Begin and End to start and stop linework
Point 6: Point Entity by itself after End code
Points 7-11: Linework by code defined as Polyline entity type; using End as break between linework
Points 12-15: Linework by code defined as Polyline entity type; using Begin as break between linework
Points 16-19: Linework by code defined as Polyline entity type; using # after code instead of Begin/End to separate linework
Points 20-22: Linework by code defined as Polyline entity type without using Begin/End to start/stop linework
Points 24-26: 3 point curve using on PC code
Points 27-30: 3+ point curve using PC/Point codes
Points 32-33: 2 point tangent curve using PC/Point codes
Points 35-39: reverse curve using PC/Point codes
Point 40: Regular point without extra descriPointion
Point 41: Using // to use a code descriPointion as a suffix
Point 42: Using \\ to use a code descriPointion as a prefix
Point 43: Using / to append a descriPointion
Point 44: Using \ to add a descriPointion as a prefix
Point 45: Using ROT and a Point# to rotate to that Point#
Point 46: Using ROT and a value to set the rotation
Point 47: Using ROT by itself to rotate to the next Point#
Point 48: Regular point without rotation
Point 49: Using AZI and DIST codes to offset the point
Point 50: Using SZ with value to set size of symbol
Points 51-52: Using SZ by itself to size symbol by the distance to the next point
Point 53: Using SZ with 2 values to draw multiple symbols at those sizes
Points 54-55: Using 2ND code to size the symbol
Points 56-58: Using 2ND and 3RD codes to size the symbol in 2 dimensions
Points 59-62: Using CLO to close the linework
Points 63-64: Using RECT with two points and a value to create a rectangle
Points 65-67: Using RECT with three points to create a rectangle
Points 68-69: Using OH to offset right a fixed amount
Points 70-73: Using OH on multiple points to offset various amounts
Points 74-75: Using multiple OH on the same point to offset polyline multiple times
Points 76-77: Using OH with negative value for offset to left
Points 78-79: Using OFL with value for offset left a fixed amount
Points 80-81: Using OFB with value to offset both left and right a fixed amount
Point 82: Using CIR to draw circle at specified radius
Points 83-84: Using CIR to draw circle using two points for center and perimeter
Points 85-89: Using CIR to draw best-fit circle through points on perimeter
Points 90-91: Using JPN to join linework to another Point#
Points 92-95: Using SMO to create smoothed linework
Points 96-97: Using JOG to create additional linework segment extensions
Points 98-102: Using GAP to create a break in the linework
Points 103-106: Using LFT to switch linetype to left side
Points 107-109: Using WALL3D with specified height value
Points 110-112: Using WALL3D with height from 2nd point
Points 113-115: Using BLOCK3D with height and three points to define parallelogram
Points 116-123: Using BLOCK3D with height and multiple points to define perimeter
Points 124-128: Using FACE3D with multiple points to make a surface

Points 129-132: Using HOLE3D with multiple points to define the perimeter of a hole in the FACE3D surface
Point 133: Using code definition with Attribute Format set to Text and only Elevation turned on with Label Decimal On Point



PointCAD Coding

Field-to-Finish supports an early Carlson style of linework coding called PointCAD. The PointCAD codes use numbers with +,-,∗ symbols as follows:
+0 Starts a regular 2D line (not a polyline) that is open.
∗0 Starts a regular 2D line that is closed.
+4 Starts a curved 2D polyline that is open.
∗4 Starts a curved 2D polyline that is closed.
+1 Begins a 3-point arc.
-0 or -1 or -3 or -4 or -5 or -6 or -7 Ends a line.
+5 Starts a 3D polyline that is open.
∗5 Starts a 3D polyline that is closed.
+6 Starts a 2D polyline that is open.
∗6 Starts a 2D polyline that is closed.
+7 starts line whose type (2D line, 2D polyline, 3D polyline) is specified by the point's field code definition. If the field code definition is to use points, then a 2D line is started.
+2 Middle point of 3 point arc
-05 starts a curved 3D polyline section.
-50 ends a curved 3D polyline section.
+8 starts a 2D and 3D polyline combination that is open.
∗8 starts a 2D and 3D polyline combination that is closed.
-8 ends a 2D and 3D polyline combination.
-08 starts a 2D and 3D polyline combination curve that is open.
-80 reverts back to a straight 2D and 3D polyline combination.



PointCAD linework coding examples

GIS Processing

With GIS processing activated, the entities created by Field-to-Finish are linked to a GIS feature name and attributes. These GIS links can be used by the routines in the GIS module such as Input-Edit GIS Data.

GIS processing in Field-to-Finish starts with the GIS Table setting in the initial Draw Field To Finish dialog. The GIS Table is the .GIS file created by the Define GIS Features command which defines the GIS feature names and attributes. Setting the GIS Table is optional but useful. The GIS Table is used as the reference in the Set functions for selecting a GIS feature name to assign to Field-to-Finish codes. Additionally, when processing the Field-to-Finish codes, any associated attributes from the GIS Table will be attached to the entities. Also, attributes generated from Field-to-Finish are added to the GIS Table. So using the GIS Table links the GIS module commands with Field-to-Finish.

Each Field-to-Finish code has settings to assign GIS feature names. In the Edit Field Code Definition dialog, the GIS Setup button brings up a dialog for setting the GIS feature names and attribute options for the current code. Since Field-to-Finish codes are capable of drawing both points and linework and GIS can have different features for points and linework, there are separate settings for the GIS feature names for points and linework. For example, a Field-to-Finish code UP for utility pole could be setup to draw both points with symbols at the poles and polylines between these points. Then you could have different GIS feature names for the pole points and linework with separate GIS attributes for each.

For Attributes to Create, these options create GIS attribute data which is stored in the database setup by the GIS Settings command and linked to the entities created by Field-to-Finish.

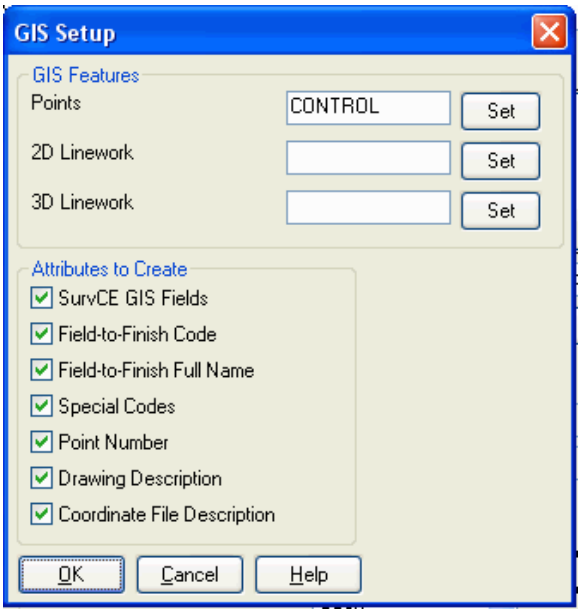
SurvCE GIS Fields: This option uses the attribute data generated by SurvCE which is stored in a .vtt file with the same file name as the current coordinate file except with the .vtt extension.

Field-to-Finish Code: This option creates an attribute named CODE with a value of the Field-to-Finish code name (ie. UP).

Field-to-Finish Full Name: This option creates an attribute named FULL_NAME with a value of the Field-to-Finish Full Name (ie. Utility Pole).

Special Codes: This option creates attributes for Field-to-Finish special codes including OH (Offset Horizontal), OV (Offset Vertical), SZ (Size), ROT (Rotation), AZI (Azimuth) and DIST (Distance).

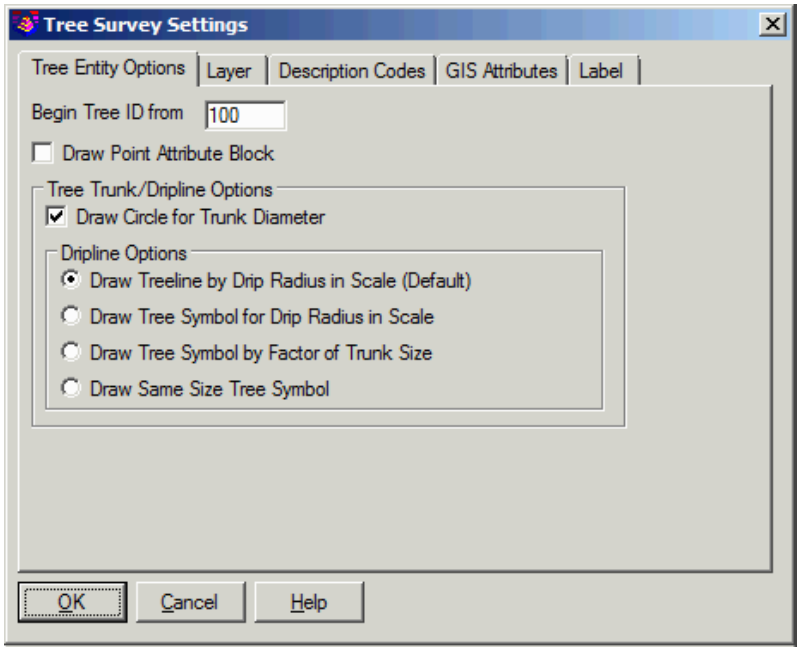
Point Number: This option creates an attribute named POINT_NAME with a value of the point number from the coordinate file.
Drawing Description: This option creates an attribute named POINT_DWG_DESC with a value of the point description for the point block created in the drawing.
Coordinate File Description: This option creates an attribute named POINT_RAW_DESC with a value of the point description from the coordinate file.



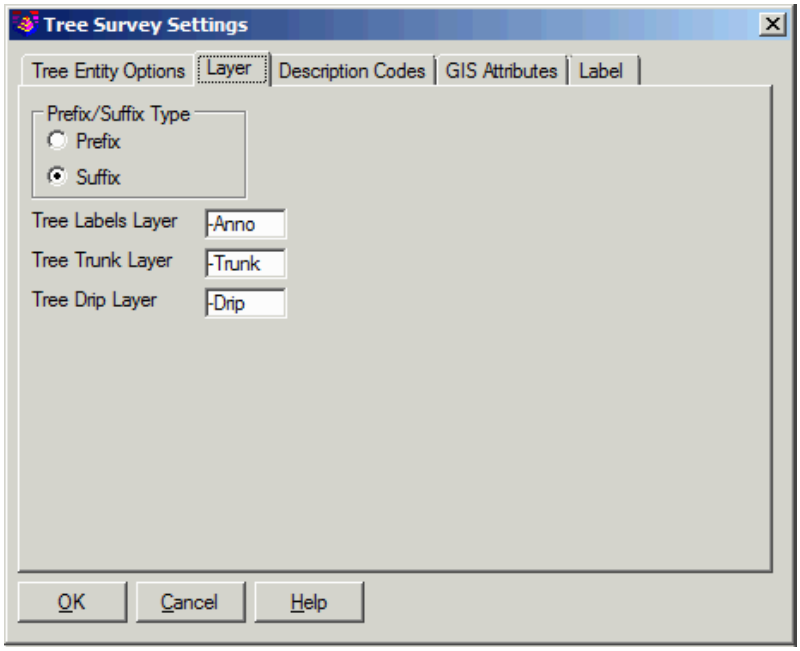
Default Code Tables
Default code tables are installed under Carlson Projects\Settings including Carlson.fld and the following DOT's: CA, CO, FL, IA, IL, IN, LA, MA, MD, MN, MO, MS, NC, ND, NE, NY, OH, SD, TX, WA and WI.

Tree Surveys
Tree surveys can be coded simply by using general Field-to-Finish coding methods such as defining a code for a tree ("OAK") with a tree symbol and using the SZ special code for sizing the symbol. For tree survey specific features, go to the Tree Survey button on the first Field-to-Finish dialog. This function brings up a dialog with tree survey settings. The tree survey works with three attributes for each tree: trunk, drip and tag. Trunk is the diameter of the tree trunk. Drip is the radius of the tree canopy. Tag is an id for the tree for reporting.

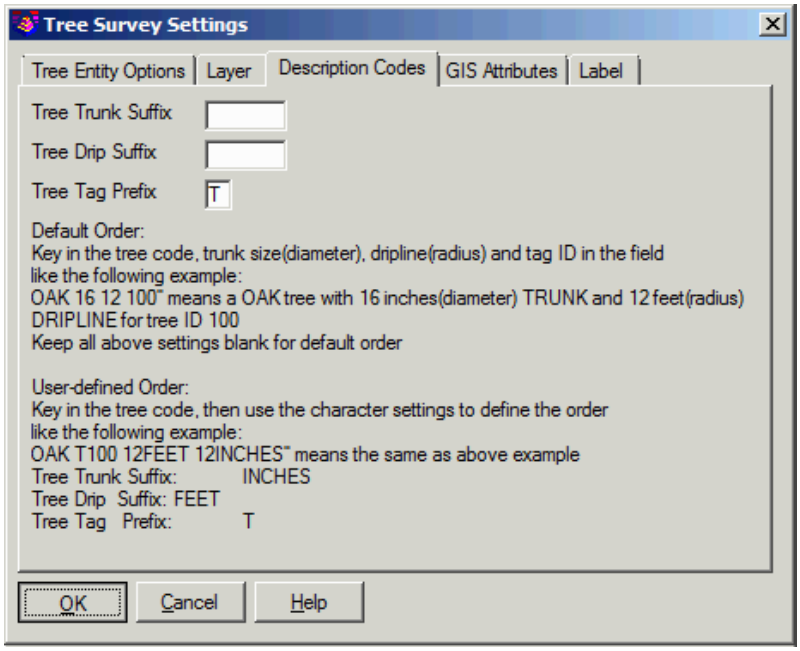
Important: The Tree Survey Settings apply to codes that are set to a Feature Type of Tree. To set the Feature Type, go to Edit Codes and then the General tab of the Edit Field Code Definition dialog.



On the Tree Entry Options dialog tab:
Begin Tree ID From: This is the number to start incrementing tree tags from in case the tree coding is missing tags and you want to assign tags for reporting.
Draw Point Attribute Block: controls whether to draw the point block with the point #, elevation and description attributes.
Draw Circle for Trunk Diameter: creates a circle with the trunk diameter.
Draw Treeline by Drip Radius in Scale: shrinkwraps the tree driplines to get the overall treeline perimeter.
Draw Tree Symbol for Drip Radius in Scale: draws individual symbols for each tree using the symbols defined in the code table and scaled by the drip size attribute.
Draw Tree Symbol by Factor of Trunk Size: draws individual symbols for each tree using the symbols defined in the code table and scaled by the trunk size attribute multiplied by 12. For example, a 10" trunk size is drawn as a 10ft symbol.
Draw Same Size Tree Symbol: draws individual symbols for each tree using the symbols defined in the code table and at size of 6.

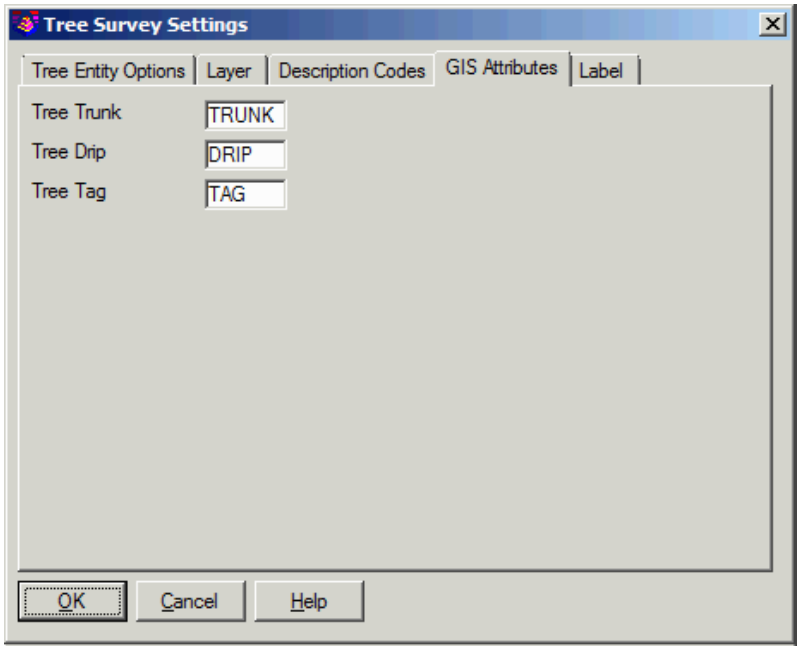


On the Layer dialog tab, there are optional layer names for different types of tree entities to append either as a prefix or suffix to the layer from the code table.

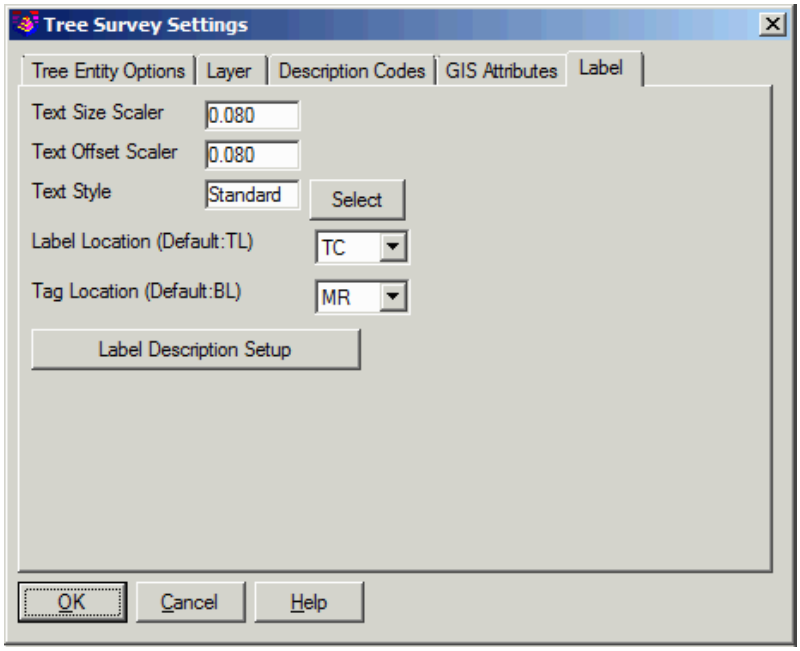


On the Description Codes tab, there are setting to help identify the tree attributes in the point description. The program looks for the trunk size, drip size and tag ID in the point description after the tree code. By default, the program expects the attributes to by in the order of trunk size, drip size and tag ID. Here's an example default order:
OAK 16 12 100
where OAK is the tree code from the code table, 16 is the trunk diameter, 12 is the drip radius and 100 is the tag ID.

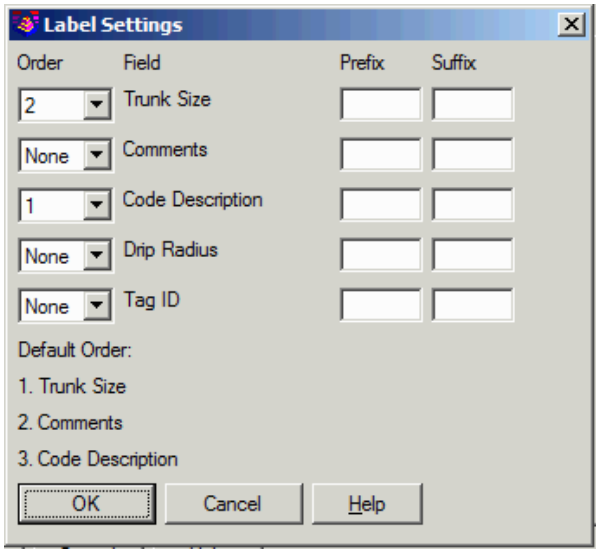
If the attributes are in a different order, then the suffix/prefix settings can be used to identify the attributes. When the program finds a specified prefix or suffix, that tells the program which attribute to use. For example, if the Trunk Suffix is "in" and the Drip Suffix is "ft" and the Tag Prefix is "T", then
OAK T100 16in 12ft
means tag ID of 100, trunk diameter 16 and drip radius 12 feet.



In addition to looking for the tree attributes in the point description, the program can also read these attributes from GIS fields. On the GIS Attributes dialog tab, you can set the GIS field names for the tree attributes.

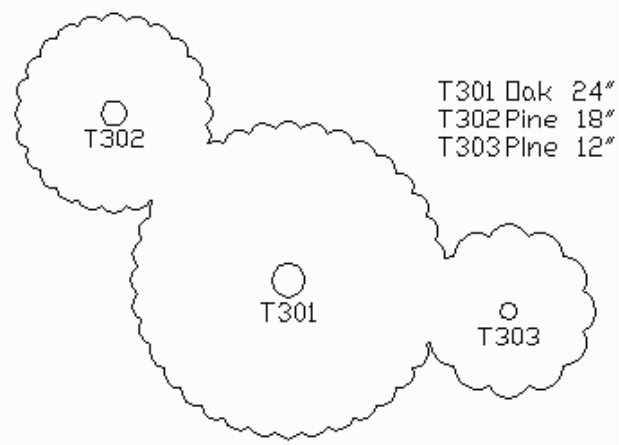


On the Label tab, there are settings for the tree text labels for the size, offset from trunk center, style and location. When creating a tree table, only the tag text is labeled. Otherwise, the label is drawn. The Label Description Setup dialog sets which fields to include the the label, the field order, prefix and suffix.



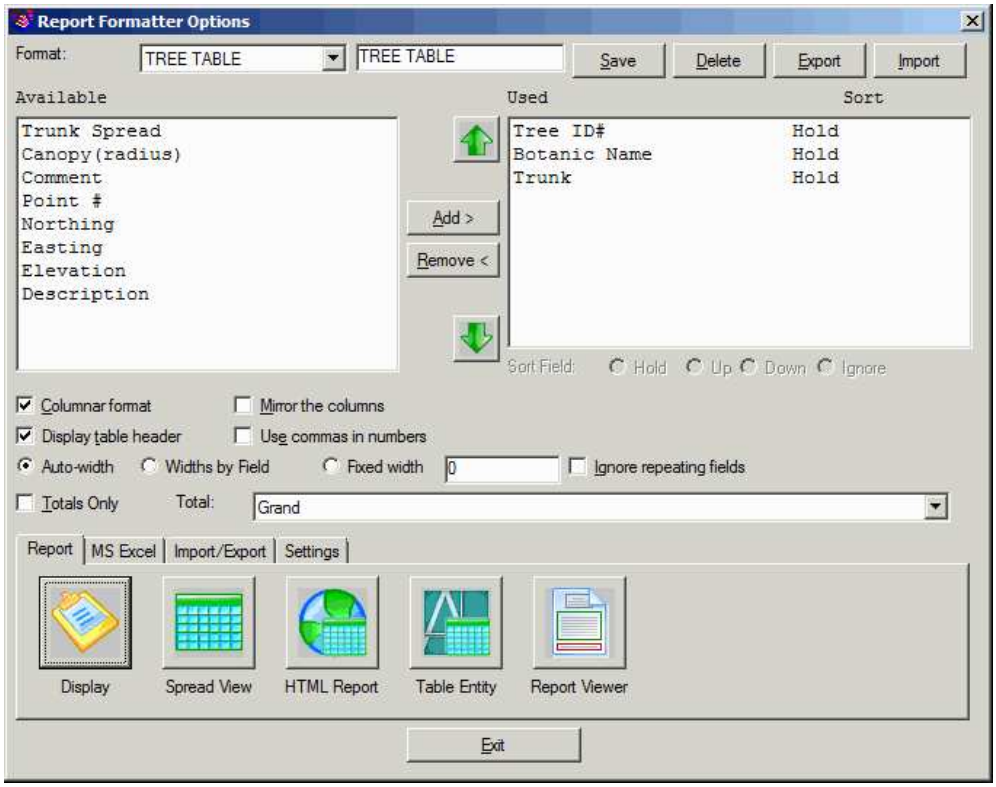
When Field-to-Finish draws entities, the program checks for codes set as tree features and applies the settings from the Tree Survey dialog. When tree features are found, the number of trees are reported along with a prompt for whether to draw a tree table. The tree table has the tag ID, code description and trunk diameter.

Here is an example with the following three points:
Point# Northing Easting Description
1 4994.73 4923.15 OAK 24 38 301
2 5034.59 4881.40 PINE 18 24 302
3 4987.32 4975.79 PINE 12 20 303



Dripline drawn as Treeline method along with a tree table.

Another feature of Tree Survey is the Tree Report under the Report Codes/Points function. The Report Formatter option can be used to make a custom report and output to Excel or create a custom table in the drawing.



Tree ID Botanic Name Trunk
T301 Oak 24"
T302 Pine 18"
T303 Pine 12"

Pulldown Menu Location: Tools > Points
Keyboard Command: fld2fin
Prerequisite: A data file of points with descriptions

List Points

This command generates a report of point numbers, northings, eastings, elevations and descriptions.

Selection Method-Range: Allows you to specify the points to list by point number range

Selection Method-Area: Allows you to select a closed polyline to list all of the points inside of that polyline.

Selection Method-Selection Set: Allows you to specify the points to list by selecting them from the drawing.

Range of Points: If you are using the Range method, specify the range of points to list here. To quickly specify all points, click the All button.

Description Match: Can be used to filter the point list. For example, entering "EP" for the Description Match would only list those points with a description of "EP". An asterisk (∗) is the default setting, it matches any character sequence, meaning no filtering occurs.

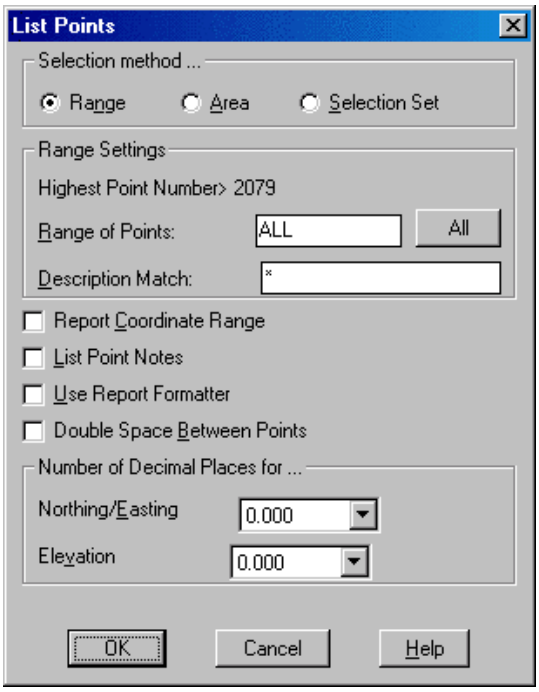
Report Coordinate Range: When checked, the point list will include the minimum and maximum northing, easting and elevation.

List Point Notes: When checked, any additional point notes assigned to the points will be included in the point list. Point notes can be entered using the *Input-Edit Point* command found in *Coordinate File Utilities*.

Use Report Formatter: When checked, you may customize the fields and layout of the point report using the Report Formatter. The Report Formatter can also be used to export the point report to Excel or Access.

Double Space Between Points: When checked, the report will be double spaced.

The point list report is displayed in the Standard Report Viewer which can print, draw and save the report file. This report viewer cannot be used to edit the coordinate file. Instead use the *Edit Points* command in the ∗Points∗ menu.



Pull-Down Menu Location: Points
Prerequisite: points in a coordinate file or on the screen
Keyboard Command: listpt

Edit Points

This command edits point data in the current coordinate file. The current coordinate file can be set with the Set Coordinate File command. Edit Points shows all the points in the coordinate file. New points can be added and points can be deleted by using the Insert and Delete keys.

This tool also lets you edit notes associated with each point. While the standard point description is limited to 32 characters, the drawing notes are not. When you click on a given point, you can add numerous lines of notes about that point in the bottom of the dialog. Keep in mind that these notes are stored in a separate file with the extension ".not" having the same name as the CRD and residing in the same folder.

Pull-Down Menu Location: Points
Keyboard Command: EDITPT
Prerequisite: None

Erase Points

This command erases AgStar points inserts from the drawing. The points to erase can either be selected from the screen or specified by point number. Erasing a AgStar point will erase the point symbol, point attributes, and point node. The points may optionally be erased from the coordinate file. As long as the points are not deleted from the coordinate file, they can be redrawn with *Draw-Locate Points*.

Prompts

Select points from screen or by point number (Screen/<Number>)? *Press Enter*

Point numbers to erase: *1-5*

Delete points from coordinate file (Yes/<No>)? *Press Enter*

Erasing AgStar Points ....

Number of points erased> 5

Pull-Down Menu Location: Points
Keyboard Command: DELPT
Prerequisite: AgStar points to be erased

Import TextASCII File

This command converts point data from an ASCII text file into the current AgStar coordinate (.CRD) file. Each line of the text file can contain any combination of point number, northing, easting, elevation and description. All point information should be on one line with the values separated by a comma, space or other delimiter. Under the Source File Format setting you can choose from some specific formats or User-Defined. For User-Defined, the format of the text file is specified in the Coordinate Order field where the value identifiers are listed with the appropriate delimiters. For example:

For a text file with northing, easting, elevation and comma delimiters:

5100.0,5150.5,485.1

5127.1,5190.3,487.3

The Coordinate Order would be:

Y,X,Z

For a text file with point number, easting, northing, elevation, description and space delimiters:

1 5000.0 5000.0 490.3 TRAV

2 5030.4 4930.5 495.5 TRAV

The Coordinate Order would be:

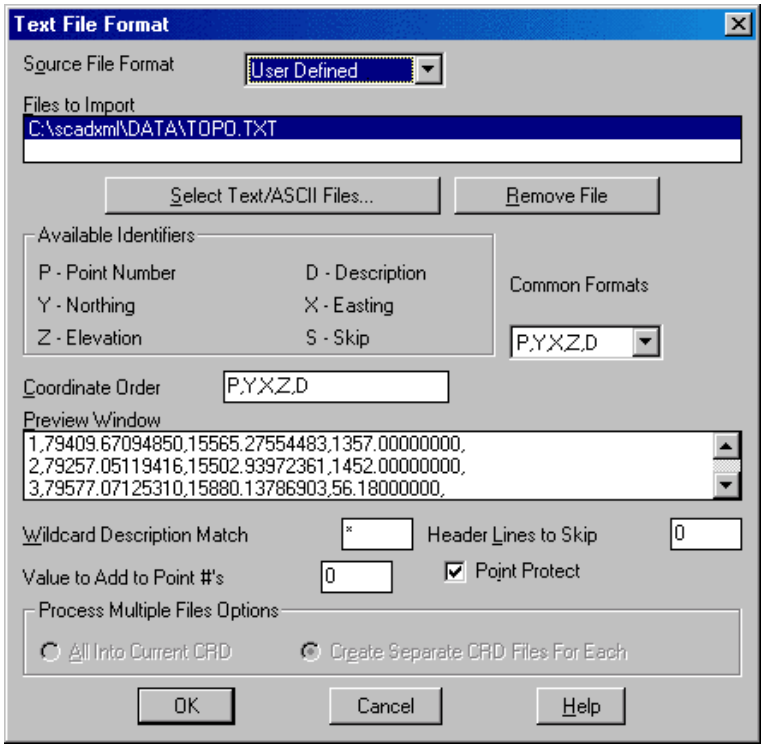
P X Y Z D

Common formats can be selected from the Common Format List. All the lines in the text file should contain only point data and any header lines should be removed. To read the text file, pick the Select Text/ASCII File button and choose the file to read. Then the selected file is displayed in the Preview Window to help with filling out the Coordinate Order. When the Coordinate Order is set, click OK to read the text file. The Wild Card Descriptions Match allows for only point with matching

descriptions to be imported. With Point Protect active, the program will check if a point number already exists in the CRD before importing the point. If a point conflict is found, you can either assign a new point number or overwrite the old point. The Value to Add to Point Numbers allows you to renumber the points as they are imported. The Header Lines to Skip value is the number of lines not to be processed at the start of the text file.

Multiple files can be imported at once. In the Select Text/ASCII Files routine, you can select multiple files by using the Shift or Ctrl keys while picking files. You can also run Select Text/ASCII Files multiple times. The files to import are listed in the top scroll display window. The point data from all the import files can be stored to the current CRD file or to separate files for each import file. The separate file option will name the CRD files by the import file with a .CRD file extension. For example, the import file job125.txt would create job125.crd.

The special formats of Leica ∗.gsi files, TDS ∗.cr5 files, Geodimeter ∗.obs/∗.raw files, Laser Atlanta ∗.txt files, Trimble ∗.pos files, Zeiss ∗.txt files, Traverse PC ∗.trv files, Maptech and Benchmark ∗.dat files can be directly imported by choosing that File Format at the top of the dialog.



Pull-Down Menu Location: Tools > Points
Prerequisite: A text file to read
Keyboard Command: readpt

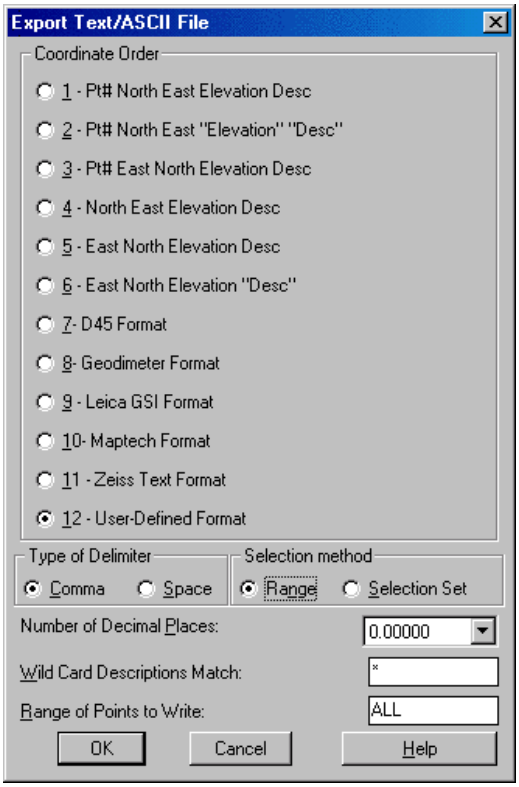
Export TextASCII File

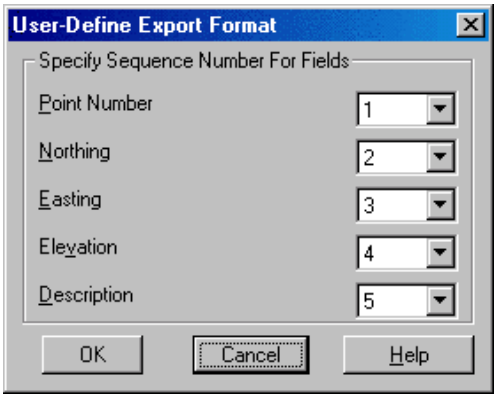
This command outputs point data from the current AgStar coordinate file to an ASCII text file. Specify the type of file to write with the Coordinate Order radio buttons. There are several variations on point number, northing, easting, elevation and descriptions as well as specific formats for Leica, Geodimeter, Zeiss, Maptech and D45 formats.

In addition there is an option, User-Defined Format, to define the order of the fields output. When using the User-Defined format, after selecting OK, the User-Define Export Format dialog will appear.

On this dialog, specify the order of the fields by defining a number sequence in each field. You can skip fields and omit data in the output file by leaving None in the sequence field for this data.

The point data to export can be specified by selecting a range of points, or by a selection set of points from the screen. The desired option is specified by selecting the option under the Selection Method on the Export Text/ASCII File dialog. A description filter is also available for exporting only points from the range or selection set with certain descriptions. After selecting the OK button, another dialog appears that allows you to specify a new text?ASCII file or to append data into an existing file. The standard file selection dialog allows you to specify the export file name.





Pull-Down Menu Location: Tools > Points
Prerequisite: A Coordinate File (.CRD)
Keyboard Command: writept

Set Coordinate File

This command allows the user to set the name of the current coordinate file. This file is used by different commands that compute, store and recall the coordinates. The file has an extension of .CRD and by default is stored in the configured data subdirectory. When prompted for the name, if you type in a path name the file will be stored in the specified path. If you don't specify a path then the default path that is configured in the *Configure Survey* command will be used.

The Existing button is the default and can be selected by pressing [Enter] (note the thicker highlighting around the button) or select either button by clicking on it with your pointing device. The underlined characters are the short cut keys that can be selected by the character and pressing [Enter].

Pull-Down Menu Location: Tools > Points
Prerequisite: None
Keyboard Command: setcrd

CooRDinate File Utilities

This command allows you to manipulate the coordinates stored in a coordinate (.CRD) file. One of the most important commands is the Update CRD File from Drawing which allows you to update the file after editing the drawing with commands such as *Erase*, *Move*, *Rotate* or *Change Elevations*. Another handy option is the *Draw Entities by Point Number* which allows the user to input point number ranges and plot Lines, Arcs, Polylines or 3D polylines. Coordinate files have either numeric or alphanumeric point numbers. Alphanumeric point numbers consist of nine or less digits and letters (i.e. point number 7A). The type of point number format is displayed at the top title bar of the dialog shown on the next page.

Coordinate File Utilities Options

Open CRD File: Allows the user to switch to another file. When you exit Coordinate File Utilities this will be the current file that you work with in AgStar.

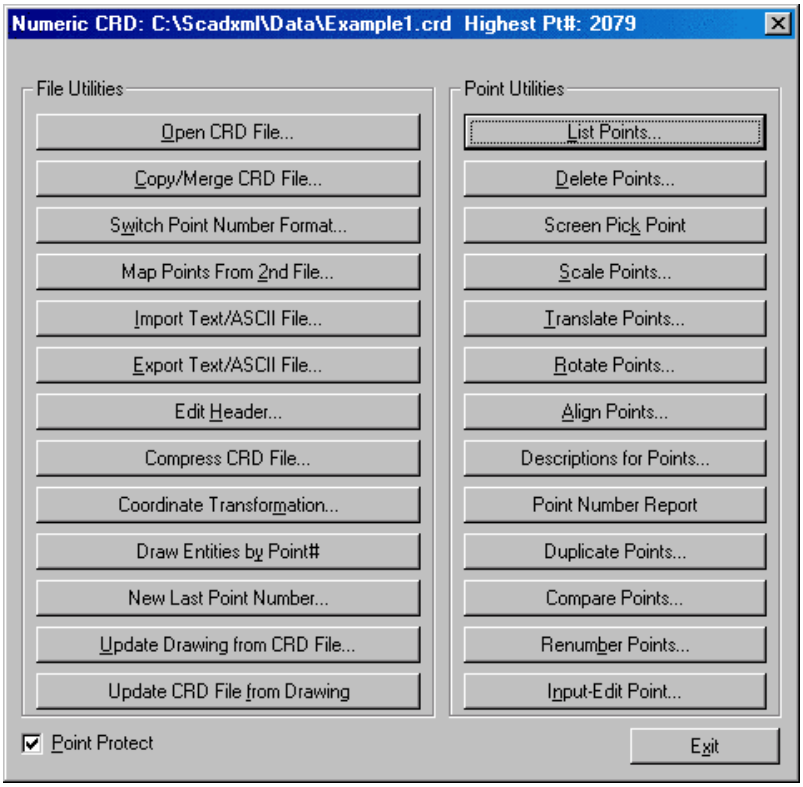
Copy CRD File: Copies a coordinate file to another file name. This can be used to make a backup of your coordinate file.

Switch Point Number Format: Allows you to convert the current CRD file from numeric format to alphanumeric format.

Import Text/ASCII File: This routine converts point data from a text file into the current coordinate (.CRD) file. See the *Import Text/ASCII File* command in this chapter for more information.

Export Text/ASCII Text File: This routine outputs point data from the current coordinate (.CRD) file to a ASCII Text file. See the *Export Text/ASCII File* command in this chapter for more information.

Edit Header: Enter or edit the job information associated with the coordinate file. Fields include Job Description, Job Number and Job Date.



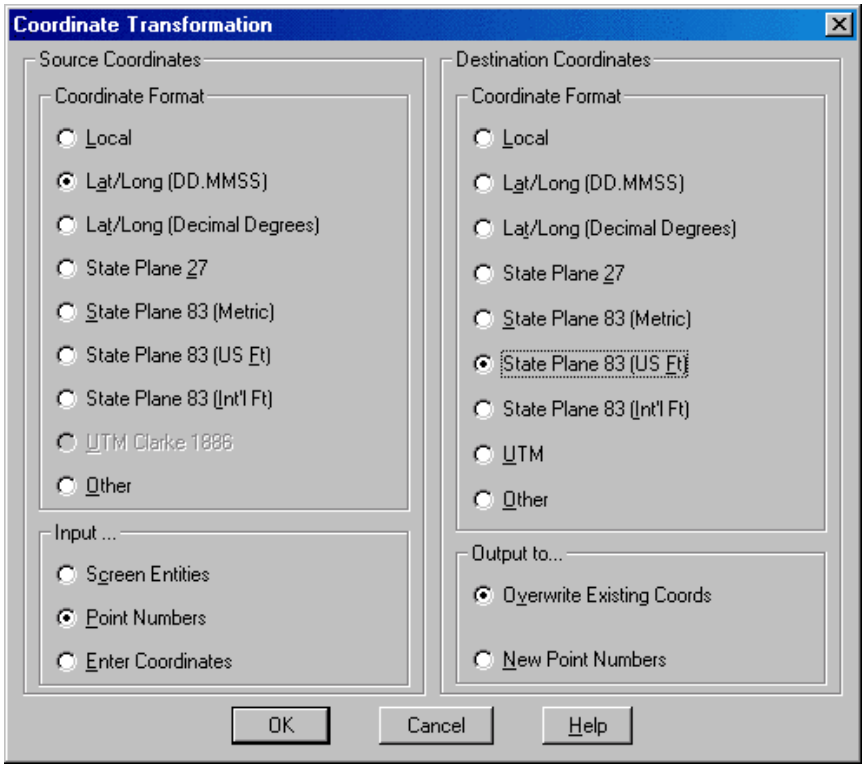
Compress CRD File: Removes unused point numbers by renumbering high point numbers into the unused spaces. For example, for an original file with points 1,2,105,107,108,109 would be compressed to 1,2,3,4,5,6.

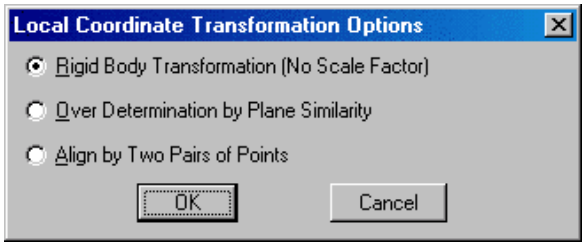
Coordinate Transformation: Transforms coordinates between local, state plane 27, state plane 83, latitude/longitude, and Universal Transverse Mercator (UTM). Works on individually entered coordinates, by range of point numbers and with on-screen entities. For converting between state plane 27 and 83, AgStar calls upon NADCON from the National Geodetic Survey to apply the latitude/longitude adjustment. The NADCON program, ndcon210.exe, is stored in the AgStar EXEC directory.

The Enter Coordinates input option transforms one coordinate at a time. The coordinates can be typed in or use the Input Point Number option. Output Point Number is an option to store the results in the coordinate file.

When transforming a local coordinate system, there are three methods as shown in this dialog. The Align by Two Pairs of Points option uses two pairs of source and destination coordinates. The first pair defines the translation as the difference between the source and destination northing and easting. This destination point is also the pivot point for rotation. Rotation can be entered directly or defined by a second pair of points where the bearing between the first and second source points is rotated to align with the bearing from the first and second destination points. There is an option to also apply scaling. The scaling holds the angle between points and adjusts the distances by the scale factor. The scale factor is calculated for each point as the elevation factor at the first source point times the grid factor at the first destination point averaged with the elevation factor at the transform point times the grid factor at the transform point.

The other local transformation options are used when there are more than two pairs for translation points. Since two pairs of points are sufficient to define the translation and rotation, more than two pairs of points provides more than enough information. Over Determination by Plane Similarity is used to find the least squares best fit transformation for all the given source and destination points. Besides doing a translation and rotation, this option will also scales the points during the transformation. The Rigid Body Transformation also does a best fit least squares transformation but applies only translation and rotation with no scale.





When running AgStar with AutoCAD Map, the Coordinate Transformation dialog has an option called "Other" which activates all the AutoCAD Map transformations.

Draw Entities by Point#'s: Draw Lines, Arcs, 3DLines, Polylines or 3DPolys by defining a range of point numbers.

Example Prompts:

Enter Menu Option? <L>: *P*

Plot Entities by Point Number

Type of entity, Arc/Polyline/3dpoly/2dline/Exit/<Line>: *P*

This response causes the program to plot polylines.

Example: '1∗4-7-10∗12-5-8' would draw lines from point number's 1 through 4 then to 7, to 10 through 12, then to 5 to 8. (limit 132 characters)

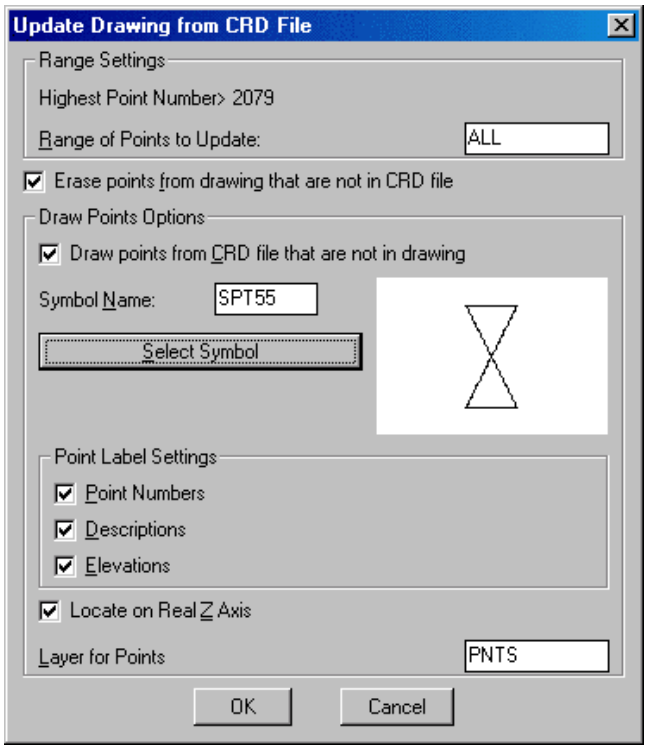
Undo/<Enter point numbers or ranges>: *1∗10-20∗30*

The program draws a polyline from point number 1 through 10 to point number 20 through 30.

New Last Point Number: This option sets the highest point number in the CRD file. All points above this number are erased.

Update Drawing from CRD File: This function updates the position of AgStar points in the drawing to match the position stored in the coordinate file. This command also has options to erase and draw points. For the erase option, point are erased from the drawing if the point number does not exist in the coordinate file. For the draw option, if a point number in the CRD file does not exist in the drawing, then this point is drawn using the settings from the dialog. The number of points modified, erased and drawn is reported at the end of the command.

Update CRD File from Drawing: This function allows you to select all or some of the points in the drawing and add or update them to the .CRD file. The points can be filtered with AutoCAD's Select Objects: selection mechanism and/or wild card matching of the point descriptions. The Update Point Descriptions option determines whether the point descriptions from the drawing will be stored to the CRD file. Use this command to update the file after a global edit such as *Move*, *Rotate*, *Renumber Points*, *Change Elevations*, *Erase*, etc. This routine directly reads Leica (Wildsoft), Softdesk, Geodimeter, InRoads, Land Development Desktop, and Eagle Point point blocks.

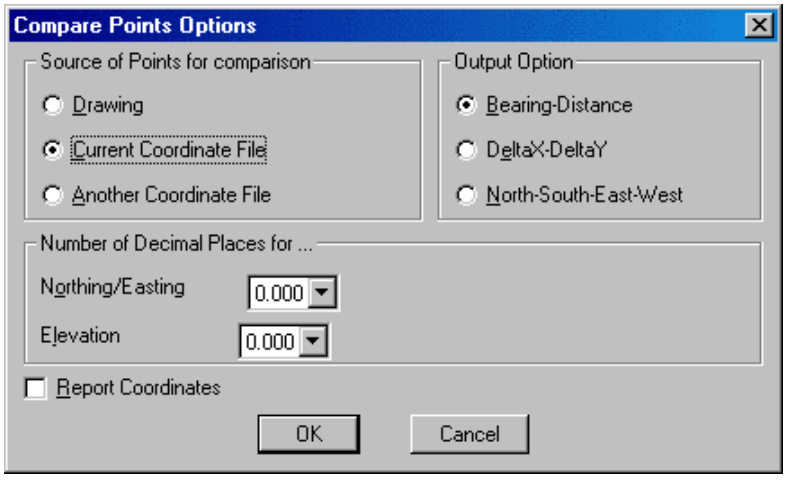


Compare Points: This function compares the coordinates in the .CRD file with either the coordinates for the matching point numbers in the drawing file, with matching point numbers from another CRD file or with different point numbers from the same CRD file. A report is created for any differences that shows the point numbers and the differences. The difference can be reported as a bearing and distance between the two points, as distance North/South and East/West or as the delta-X and delta-Y. There is an option whether to include the point coordinates in the report.

List Points: List the points stored in the .CRD file. See the *List Points* command in this chapter for more information.

Delete Points: Deletes points in the file by defining a range of point numbers to delete or by picking a polyline that defines a perimeter from which points inside or outside the perimeter are deleted.

Screen Pick Point: Pick a point on the graphics screen and add or update it's coordinate values to the .CRD file. This command does not plot a point, point attributes or point symbol. Use the command *Draw-Locate Points* command to do this.



Scale Points: This option multiplies the point northing, easting, and elevation by the scale conversion factor. You can use this routine for metric-english conversion. See the *Scale Points* command in this chapter for more information.

Translate Points: This option translates a range of points based on entered delta x and delta y, entered coordinates or translation point numbers. See the *Translate Points* command in this chapter for more information.

Rotate Points: This option rotates a range of points based on entered degrees or rotation, entered azimuths, entered bearings or rotation point numbers. See the *Rotate Points* command in this chapter for more information.

Align Points: This option does a translate based on a source point and destination point and then rotates to align the first source point and a second source point with the first destination point and a second destination point. See the *Align Points* command in this chapter for more information.

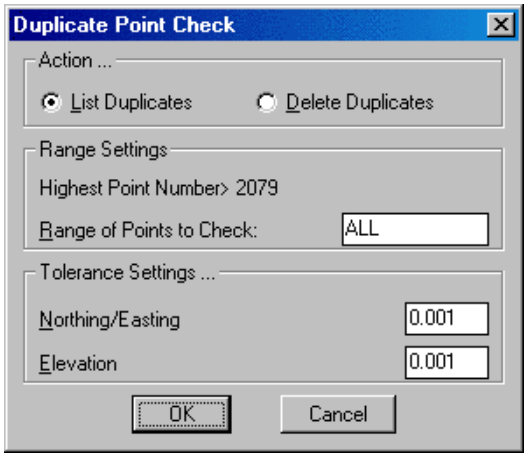
Duplicate Points: This function searches the CRD file for points with the same northing, easting and elevation. The tolerances for considering points to have the same coordinate are set in the dialog separately for northing/easting and elevation. To be counted the same coordinate, both the northing/easting and elevation must be within the tolerance distance. The duplicate points can be erased or only reported. For the erase option, the first point number is kept and any higher point numbers with duplicate coordinates are erased from the CRD file.

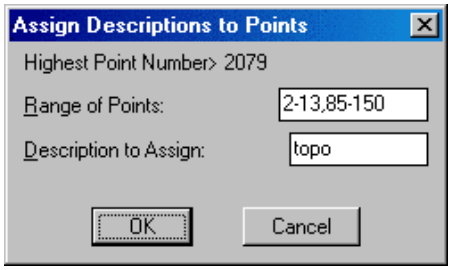
Point Number Report: This routine list the used and unused point numbers in the .CRD file.

Renumber Points: This option renumbers points in the user-specified range starting from a new point number. The old point numbers are erased. The condense points will renumber such that there are no unused point numbers in the renumbered range. Otherwise the spaces between the points is maintained. In the example shown, renumbering 1-25 with points 1,2,24,25 to starting point number 101 will result in points 101,102,103,104 if condense is on or 101,102,124,125 if condense is off.

Description for Points: This option sets the point description field with the user-specified text for a range of point numbers.

Input-Edit Point: Enter or edit the coordinate values or the description of a point. The Notes section is for adding optional point notes which are additional point descriptions. The standard description field is limited to 32 characters. Under notes, any number of lines of text can be assigned to the point. A list box shows the lines of notes. To add a note line, pick a blank line in the list box and then type in the note in the edit box belong the list box and press Enter. To edit a note, highlight the line in the list box and edit the text in the edit box.





Map Points from 2nd File: This routine adds point to the current CRD file from points stored in a second CRD file. The points to copy are specified by numbers one at a time.

Destination Point Number: *55* This is the point number to create in the current CRD file.

Source Point Number: *25* This is the point number to copy from the second CRD file.

Point# Northing Easting Elevation

25 52.516 13.328 0.000

Destination Point Number: *Press Enter to end*

Point Protect Toggle: Toggles point protection on and off.

Pull-Down Menu Location: Tools
Keyboard Command: CFU
Prerequisite: None

Create Design Surface Points

This command creates points on the design surface grid. The points are drawn in the drawing and stored in the current coordinate file. At the first prompt, you can pick inside a design subdivision to process, or choose to process the entire field.

These design points can be used for stakeout or reporting.

Prompts

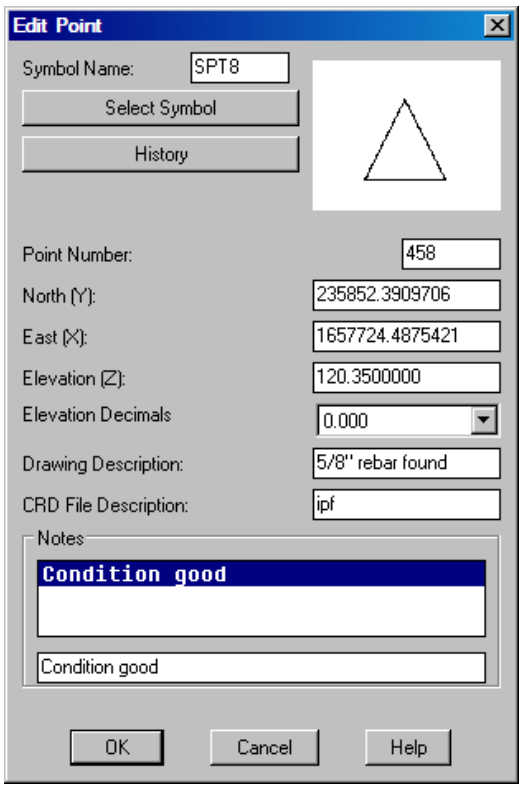
Pick inside subdivision area to select, or press <Enter> to select entire field: *pick point*
Highest Point Number: 3949
Starting Point Number <3950>: *press Enter*

Pull-Down Menu Location: Tools > Points
Prerequisite: A design field

Keyboard Command: autopnt_agstar

Edit Point Attributes

This command will edit the attributes of a Carlson point, such as the symbol type, point number, elevation and description. When this command is invoked, the command line will prompt the user: **Select point to edit (Enter to end).** At this point, you can select any part of the point including the symbol, elevation, point number or the description. Next, a dialog will appear as shown.



To change the symbol, either type in a new symbol name in the edit box, or choose the "Select Symbol" button where you can choose from a list of symbols. To change any of the other properties of the point, simply change or replace the contents of the edit box with the new information. Both Drawing Description and CRD File Descriptions are displayed. When a change to the Drawing description is made, this change will not be reflected in the coordinate file. This allows the change of a description that is defined in the Field to Finish (fld) table for a particular code. If a change is made in the CRD File description, it will be reflected in the coordinate file. Take note that if the CRD file description is changed, running Field to Finish will change the definitions for the point(s) changed. If you change the point number to a number that already exists in the current CRD file, and point protect is ON, you will be prompted **[O]verwrite w/new coordinates, overwrite [A]ll, or use number <1000>:.** You can choose to use the next available point number in the CRD file (this is the default), or overwrite the point number. The properties that you modify, with the exception of Drawing Description, will update the current CRD file. All modifications will update screen entities. Selecting the History button will bring up another dialog box that displays the point history of the point chosen. A history of the point will be listed, but only if, under General Setting, the Maintain CRD History File had been set to ON (selected) for the coordinate file that you are working with. With the CRD History feature of Carlson, all point changes can be rolled back.

You may also choose to use the AutoCAD *DDATTE* command to change the attributes of a point. If you do this, then the CRD file will not be updated and if you change the elevation attribute, the point will not change its current Z location.

Pulldown Menu Location: Tools > Points
Keyboard Command: editpnt
Prerequisite: Carlson points

Move Point Attributes

This command allows the user to move Carlson point attributes (including the point number, elevation or description) one at a time.

Prompts

Select Point Number, Elevation, or Description to Move: *select point attribute*

Pick new location: *pick point*

Pick new angle: *pick new angle or press Enter*

Pulldown Menu Location: Tools > Points
Keyboard Command: movepnt
Prerequisite: Carlson points

Resize Point Attributes

This command sets the size of the selected point attributes (point number, elevation, description) and point symbols. This command is similar to Scale Point Attributes, but instead of scaling the size by a factor, all the select points are set to the same specified size. Points can also be chosen based upon Point Groups.

Prompts

Enter point attribute and symbol size <4.0>: *press Enter*
Scale symbols only, point labels only or both [Symbols/Labels/<Both>]? *press Enter*
Select points from screen, group or by point number [<Screen>/Group/Number]? *press Enter*
Select Carlson Software points.
Select objects: *pick the point entities*
Finding Carlson Software Point Attributes ....
Number of entities changed> 10

Pulldown Menu Location: Tools > Points
Keyboard Command: sizepnt
Prerequisite: Carlson points

Erase Point Attributes

This command allows you to erase point attributes like the number, elevation or description individually by picking on the attribute to erase.

Prompts

Select Point No., Elev, or Desc to Erase: *select point attribute*

Pulldown Menu Location: Tools > Points
Keyboard Command: erasepnt

Prerequisite: Carlson points

Inverse

This command returns/inverses the bearing/azimuth and horizontal distance between two points. The command prompts for series of points. Use the appropriate object snap mode to select the points from the screen or use the point numbers to reference coordinates stored in the current coordinate file. The results are then displayed. This command is also used in conjunction with the *Traverse* and *Sideshot* commands to occupy and backsight two points. The last two points you Inverse to are the Backsight and the Occupied point for the *Traverse* and *Sideshot* commands. Press [Enter] at the point prompt to end the command.

You can also inverse around an arc by inversing to the PC and then entering *A* for Arc option. The program will ask for the radius point, the curve direction left or right and the PT point. The curve data is then reported. There is an unequal PC-Radius and PT-Radius distance check. The tolerance for this is set in the Area Label Defaults command.

There are several input options for Inverse that are set by entering *O* for Options. Sideshot inverse holds the current occupied point and calculates the bearing/distance to each entered point. The Pairs option reports the bearing/distance between pairs of points and not for every entered point. For example, if points 1,2,11,12 were entered, the bearing/distance would be reported for 1,2 and 11,12 but not 2,11. The Auto Increment option uses the next point number by just pressing Enter. To exit the routine with Auto Increment active, *End* must be entered.

There are also several angle output options that are set at the second prompt in the Options. The angle can be reported as either Bearing, Azimuth or Angle Right.

Prompts

Calculate Bearing & Distance from starting point?

Traverse/Sideshot/Options/Arc/Point number or pick point: *(pick point)*

Traverse/Sideshot/Options/Arc/Point number or pick point: *9* Use point number 9.

PtNo. North(y) East(x) Elev(z) Description

9 4909.25 4648.37 0.00

Bearing: N 81d8'54" E Azimuth: 81d8'54"

Horizontal Distance: 261.17407461

Pull-Down Menu Location: Tools > COGO
Prerequisite: None

Occupy Point

This command sets the occupied point and backsight angle for other COGO commands such as Traverse. For setting the occupied point, you have the option of picking a point on the screen, entering coordinates at the command line or typing in a point number that will be read from the current coordinate file. Four options are available for determining the backsight direction: Azimuth, Bearing, Point and None. For the "Point" option, you may pick a point on the screen, input coordinates, or type a point number that will be read from the current coordinate file. For the "Azimuth" and "Bearing" option, you enter the backsight angle in the selected format. The "None" option sets the backsight to an azimuth of 0 (north).

You can also set the occupied point by using the Inverse command. If you inverse from point 3 to point 1, you have set point 1 as the occupied point and point 3 as the backsight. For more information, see the Inverse command.

The current occupied point and backsight are shown in the lower right hand corner of the AutoCAD status bar just below the command line.

Prompts

Set Occupied Point

Pick point or point number: *pick a point (5000 5000 0.0)*

Set backsight method [Azimuth/Bearing/None/<Point>]? Press Enter to accept the "Point" default value. To select a different option, enter the first letter of the desired option and press enter.

Set Backsight Point

Pick point or point number: *pick a point (5184.76 5381.3 0.0)*

Pull-Down Menu Location: Tools > Cogo
Prerequisite: None

Traverse

This command allows the user to input any combination of turned angles, azimuths or bearings to define a traverse or figure. The command prompts for an Angle-Bearing Code which defines the angle or bearing type. Codes 1 through 4 define the bearing quadrants; 1 being North-East, 2 South-East, 3 South-West, and 4 North-West. Code 5 is a north based azimuth, 6 an angle turned to the left, 7 an angle turned to the right, 8 a deflection angle left and 9 a deflection angle right. The command draws lines between located points (if the Line On/Off is set to on) and plots the points calculated and stores them in the current CooRDinate File if point numbering is On. If Point Protect is turned On, *Traverse* checks if the point numbers are already stored in the file. This command always occupies the last point it calculated and backsights the point before that.

There are Angle-Bearing code input options for Traverse that are set by entering *O* for Options. The Angle Right option prompts for the angle right and skips the angle-bearing code prompt. The Azimuth option prompts for the azimuth and skips the angle-bearing code prompt.

Prompts

Occupied Point ?

Pick point or point number: *(pick point)*

You will only be prompted for the occupied point the first time you use the command.

Use the *Inverse* command to set the occupied and backsight points.

Exit/Options/Line/Side Shot/Inverse/<Angle-Bearing Code <7>>: *[Enter]*

Pressing [Enter] uses the default angle right code.

Enter Angle (dd.mmss) <90.0000>: *88.1324*

You can also enter L or R to define an angle 90 degrees Left or Right.

Backsight Point ?

Pick point or point number: *(pick point)*

Number inverse/<Distance>: *100*

∗.CRD File to process <c:/sc/data/LOT.crd>: *[Enter]*

This prompt comes up only if you have not set a current CooRDinate file with another command.

Exit/Options/Line/Side Shot/Inverse/<Angle-Bearing Code <7>>>: *14∗9-45.2045*

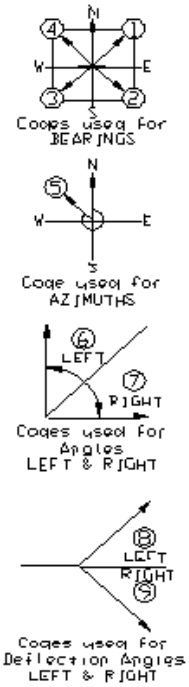
Uses the bearing defined by point numbers 14 & 9 and subtracts the angle 45 degrees, 20 minutes, and 45 seconds. You can use a + or - in this type of entry.

Number inverse/<Distance>: *N*

Point number inverse (i.e. 10*20): *14*9/2*

Causes the command to recall the distance from point number 14 to 9 and divide it by 2.

Exit/Options/Line/Side Shot/Inverse/<Angle-Bearing Code



<7>>: *L*

Select Line or Polyline that defines Bearing: *(select line that defines bearing)*

Number inverse/<Distance>: *100*

Exit/Options/Line/Side Shot/Inverse/<Angle-Bearing Code <7>>: *E*

Input an E to end the command. Enter S to execute the Side Shots command or I to execute the Inverse command.

Pull-Down Menu Location: Tools > COGO
Prerequisite: None

Side Shots

This command allows the user to input any combination of turned angles, azimuths or bearings to define a traverse or figure. The command prompts for an Angle-Bearing Code which defines the angle or bearing type. Codes 1 through 4 define the bearing quadrants; 1 being North-East, 2 South-East, 3 South-West, and 4 North-West. Code 5 is a north based azimuth, 6 an angle turned to the left, 7 an angled turned to the right, 8 a deflection angle left and 9 a deflection angle right. The command plots the points calculated and stores them in the current CooRDinate File if point numbering is On. If Point Protect is turned On, *Side Shots* checks if the point numbers are already stored in the file. All points calculated radiate from the occupied point. Use the *Inverse* command explained previously to define the occupied and backsight points.

Prompts

Exit/Line/Traverse/Inverse/<Angle-Bearing Code <7>: *6* Code 6 for angle turned to left.

Enter Angle (dd.mmss) <45.5413>: *22.3524* Angle of 22 degrees, 35 minutes, 24 seconds.

Number inverse/<Distance>: *120.91*

Enter Vertical Angle (dd.mmss) <90.0000>: *88.2548*

This prompt only comes up if you have Vertical angle prompting set to 1 or 2.

Instrument Height <5.0>: *5.12*

Rod-Target Height <5.0>: *5.12*

These prompts only come up if you have Instrument and Rod height prompting turned on.

Hz Distance> 120.86

Enter Point Elevation <1033.31>: *[Enter]* Press [Enter] to use this elevation calculated by the command.

Enter point description: *Topo Shot*

Exit/Line/Traverse/Inverse/<Angle-Bearing Code <6>>: *E*

Pull-Down Menu Location: Tools > COGO
Prerequisite: None

EnterAssign Point

This command creates a point at the user-entered coordinates. The point is both stored to the current CRD file and drawn on the screen. The program will prompt for the northing and easting. Whether the program prompts for point number, elevation and description depends on the settings in the Point Defaults command. Point Defaults also sets the point symbol and layer.

Prompts

Enter North(y): *5000*

Enter East(x): *5000*

Enter Point Elevation <>: *100*

Enter point description <>: *START*

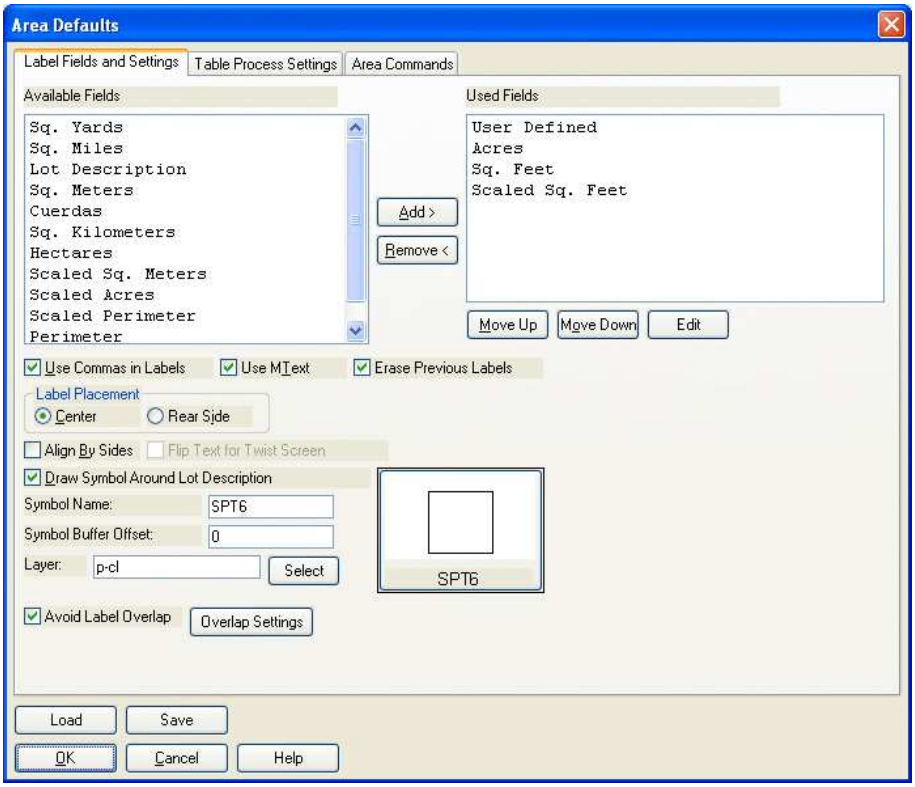
Enter North(y): *Press Enter to end the routine*

Pull-Down Menu Location: Tools > Cogo
Prerequisite: None

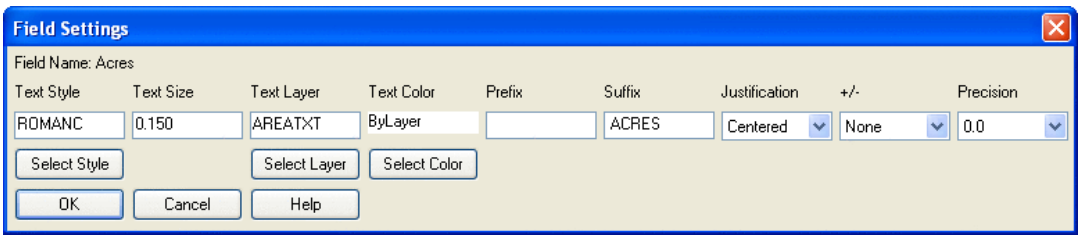
Area Defaults

This command allows you to specify default settings for area labeling. The Area Defaults dialog is divided into 3 tabs. The first is the Label Fields and Settings tab. The top portion of the Label Fields and Settings tab contains two listboxes which are used to control which of the possible ten area fields will be used for area labeling. You use the Add and Remove buttons to control which fields will be included in area labels. You can also add to the Used Fields list by double-clicking on items in the Available Fields list. The area label will include the values in the order as specified in the Used Fields listbox. To change the order you use the Move Up and Move Down buttons.

When a grid projection is defined in Drawing Setup, the Available Fields with include geodetic areas where the areas are adjusted by the projection. The Base Z from Drawing Setup is used for the elevation factor for this adjustment.

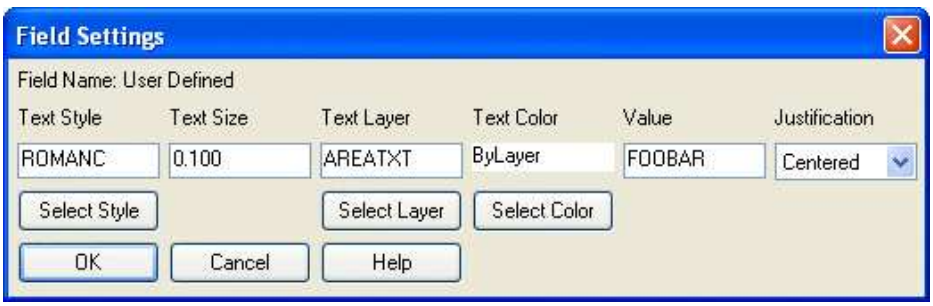


Field Settings Dialog: To control the appearance of the fields in the drawing, use the Edit button to edit the highlighted item in the Used Fields list, or double click on a field in the same list. This will call up the Field Settings Dialog.



User Defined: The Field "User Defined" can be added to place a custom fixed label in all areas. To control the value and appearance of the custom label in the drawing, use the Edit button to edit the "User Defined" item in the Used Fields list, or double click on a field in the same list. This will call up the Field Settings Dialog. In this case the "Value" setting becomes the custom label.

Scaled labels: The "Scaled Sq. Feet", "Scaled Sq. Meters", "Scaled Acres" and "Scaled Permiter" fields can be used to include area labels that are scaled based on Drawing Setup "Report Scale Factor".



Text Style: This allows you to set a text style for the area labels. You can enter the name manually or use the Select Style button to call up a dialog which presents a list of known text styles.
Text Size: This value is multiplied by the horizontal scale to obtain the actual text size.
Text Layer: This allows you to assign a layer for the area text. You can enter the name manually or use the Select Layer button to call up a dialog which presents a list of known layers.
Text Color: This allows you to to assign a color for the area text. Use the Select Color button to call up the standard color picker dialog. To use the default for the Text Layer, select ByLayer.
Prefix and Suffix: Although most area labeling uses the suffix, as in 1.25 Acres or 3.515 Hectares. But for those who prefer a prefix, as in Ac: 1.25, this routine can create that area labeling style automatically (see below for example of results of using a prefix with square feet and acres).
Justification: Use this to control whether the label field is left, centered or right justified.
+/-: This allows you to display + or - in the Prefix or Suffix of the area labels, or choose None.
Precision: Choose precision level for the currently selected field.

Below the Available and Used Fields lists the following items for further controlling area label generation:

Use Commas in Labels: This allows you to use commas in the area labels.
Use MText: Check this box to turn on the use of MText for area labels. If this is checked all area labels will be grouped into as few MText entities as possible. Area labels with different text styles, justification or layers will not be combined into the same MText entity.
Erase Previous Labels: When checked, previous area labels for the area being relabeled will be erased.
Label Placement: When auto placement of area labels is used, the labels can be placed either at the centroid of area or at the rear side. This is accomplished by selecting either the Center or Rear Side radio button, respectively. When Center is selected the user can choose to have the labels oriented according to the side lines of the area by checking the Align By Sides checkbox. When either Align By Sides or Rear Side is selected, the checkbox Flip Text for Twist Screen can be selected to have the label rotated 180 degrees to present it in the best reading orientation relative to the current Twist Screen rotation setting.
Draw Symbol Around Lot Description: When the Lot Description field is included in the Used Fields list, the user can check this checkbox to have a symbol drawn around the Lot Description field. When this box is checked, you specify the symbol name in the **Symbol Name** field or click on the current symbol (drawn to the right) to graphically choose the desired symbol. You specify the layer by entering the name in the **Layer** box or by clicking on the **Select** button to choose from a dialog that presents all known layers.
Symbol Buffer Offset: By default, the symbol will be automatically scaled according to the text length and size of the Lot Description value for the area. For additional control of symbol scaling, the user can enter a number in text size units in the Symbol Buffer Offset box. This value will be added to the automatically generated default scaling value.
Avoid Label Overlap: If this box is checked the area labels will be checked for overlaps after they are generated. Please see the Overlap Manager documentation for more information.
Overlap Settings: Click this button to go to the Avoid Label Overlaps dialog where you can review or modify the Overlap Manager settings. Please see the Overlap Manager documentation for more information.

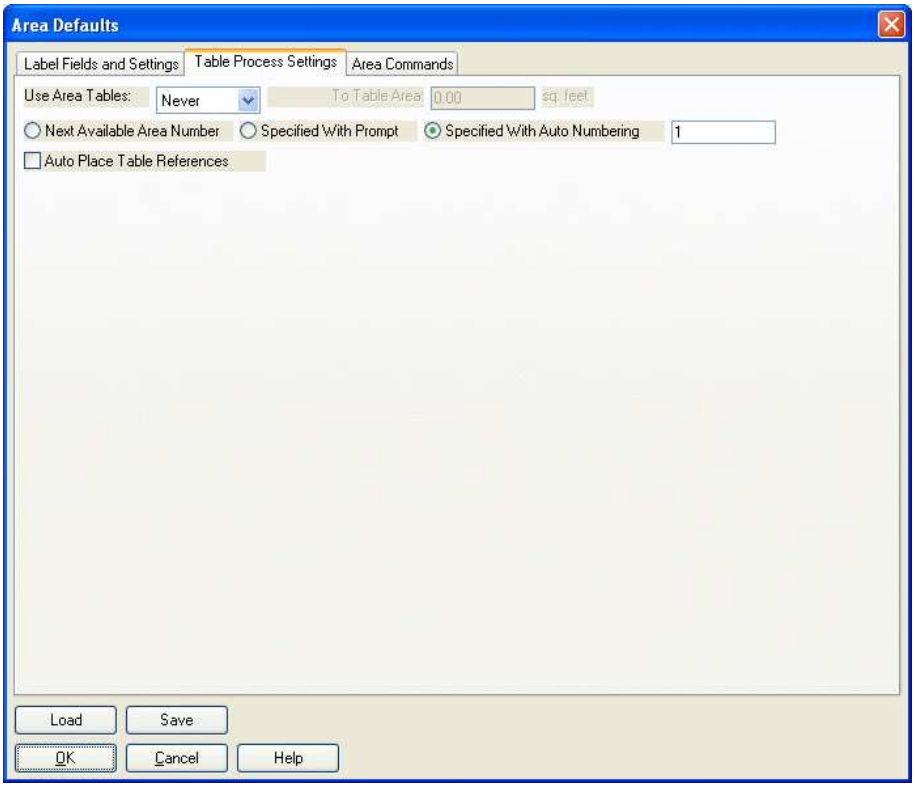


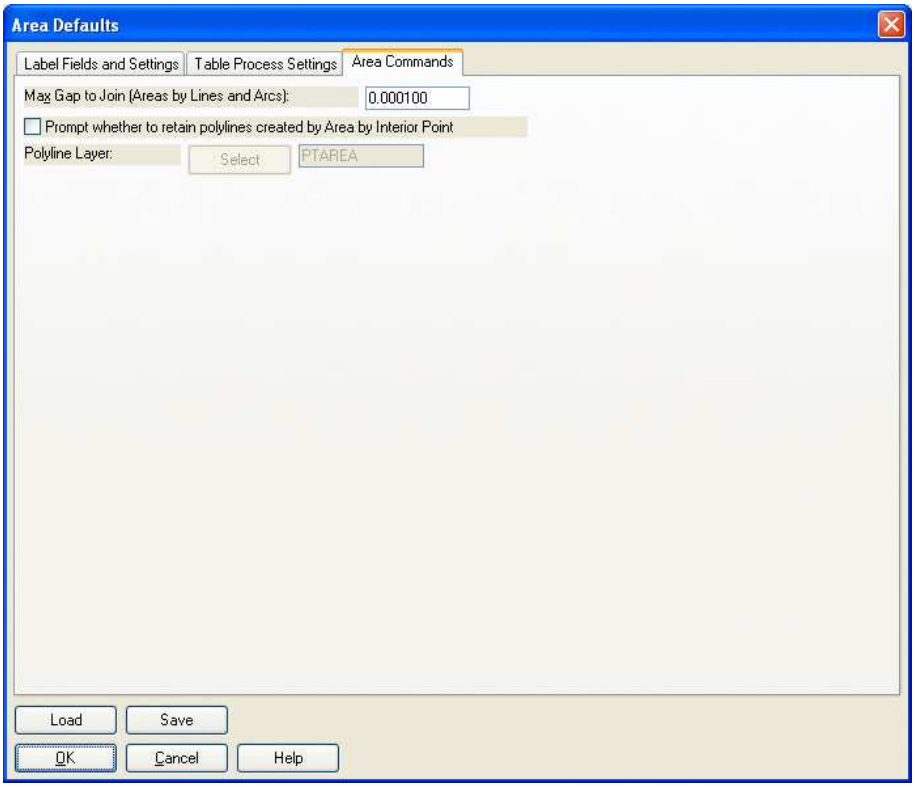
Table Process Settings Tab:

Use Area Tables: Use this control to determine whether area labels are sent to a table or not. Options are "Never", "Always" or "By Scaler".

To Table Area: When the user has selected "By Scaler" in the "Use Area Tables" list this item is enabled. When "By Scaler" is selected and the area is less than this minimum, the area label is sent to a table.

Area Reference Numbering: There are three different methods for setting the reference number: **Next Available** will automatically use the lowest available number. **Specified With Prompt** will prompt you for a number for each area. **Specified with Auto Numbering** will automatically use the lowest available number starting with the specified number.

Auto Place Table References: When checked, will automatically place the area reference label according to the settings for the area labels as specified in the Label Field and Settings tab (see above). Otherwise you will be prompted to pick each label location manually.



Area Commands Tab:
Max gap to join: You use this option during *Area by Lines & Arcs* command. When connecting lines and arcs that define the perimeter, the program will join endpoints if the distance between the two points is less than the specified gap. Otherwise the program will report an error and will not report an area.
Prompt whether to retain polylines created by Area by Interior Point: When checked the user will be asked whether to retain the polylines created by the "Area by Interior Point" command.
Polyline Layer: Will be enabled when "Prompt whether to retain polylines created by Area by Interior Point" is checked to allow the user to select the layer that any such created polylines will be placed in.

Load/Save: These buttons save and recall all the Area Default settings to a .ARS settings file.

Tip: Keep in mind that changes in Area Defaults, if changed from the Area/Layout pulldown menu, only apply to that work session. If changed within the Configure command, the changes apply to all new work sessions as well.

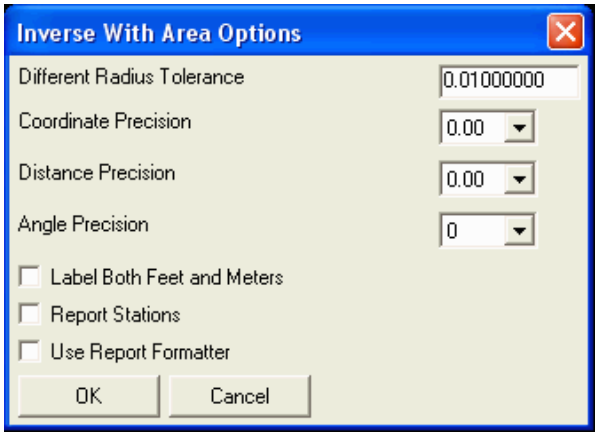
Pulldown Menu Location: Tools > Areas
Keyboard Command: defarea
Prerequisite: None

Inverse with Area

This command generates a report of the angle and horizontal distance between a series of points, and calculates the area of the closed figure defined by the points. Curve data can also be entered and reported. The points can be either picked on the screen, or entered by point number. You can also enter a range of point numbers (i.e. 1-9). The closure is reported using the total distance inversed, and the difference between the starting and ending points, as the closure error.

At the first command prompt, you can enter O for Options to bring up the command options. The Different Radius Tolerance checks that the distance between the PC and radius point and the PT and radius point match for curves. There is an option to report the distances in both feet and meters. The Use Report Formatter chooses between the

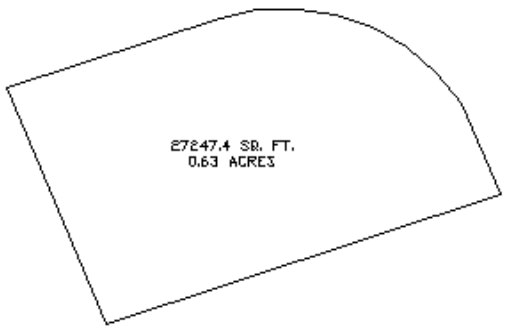
standard report or customizing the report. You can also set the decimal precisions for the report and whether to report stations for the distances along the perimeter.



The area can be labeled in the drawing using the settings from the *Area Defaults* command. If you don't want to label the area, press Enter at the pick label point prompt. This command creates a polyline of the figure which can be erased or kept in the drawing.

Prompts

Options/<Pick Starting point or point number>: *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *R* for radius
Radius point number or pick point: *pick a point*
Curve direction [Left/<Right>]? *press Enter*
Pick End of Arc or point number (U-Undo,Enter to end): *pick a point*
Pick point or point numbers (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Point number (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Point number (R-RadiusPt,U-Undo,Enter to end): *pick a point*
Point number (R-RadiusPt,U-Undo,Enter to end): *press Enter*
SQ. FEET: 27247.4 SQ. YARDS: 3027.5 SQ. MILES: 0.0
ACRES: 0.63 PERIMETER: 668.35
Pick area label centering point: *pick a point*
Erase Polyline Yes/No <Yes>: *press Enter* The command plots a polyline that represents the figure you defined if you want to keep the polyline respond with No.




```
Inverse with Area
CRD File> c:\data\newplat.crd
PNTNO  BEARING       DISTANCE  NORTHING  EASTING  STATION  DESC
903                            4940.73   2490.40  0.00     StartPt        S 48°43'58'' W 136.21  904
Closure Error Distance> 0.0000
```

```
Total Distance Inversed> 1008.07
AREA: 74664.6 SQ METERS
```

Pulldown Menu Location: Tools > Areas
Keyboard Command: ia
Prerequisite: None

Area by Interior Point

This command calculates and labels the area of the perimeter surrounding a picked interior point. The *Boundary Polyline* command is used to find the perimeter. Generally, this command will only work on closed or overlapping objects. Use *Area by Lines & Arcs* for other applications. The settings for the area label and for whether to prompt to create a closed polyline for the area are under the *Area Defaults* command.

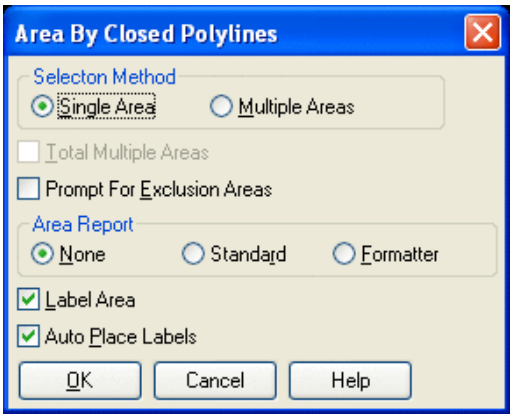
Prompts

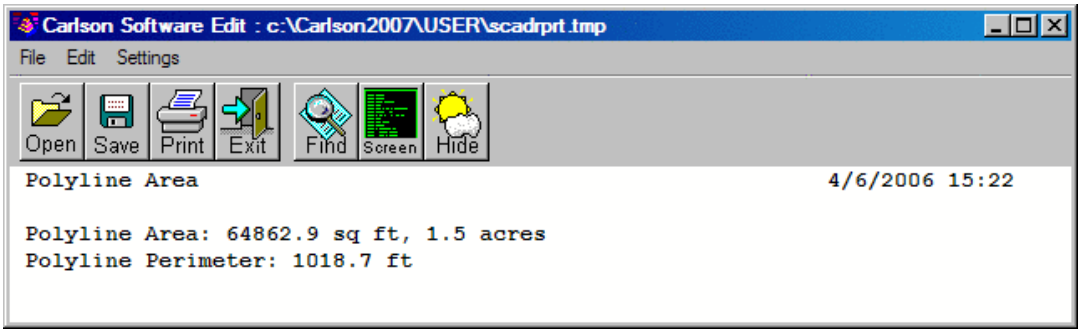
Pick point inside area perimeter: *pick a point*
Pick area label centering point (Enter for none): *pick a point*
The area is then plotted at the point selected.

Pulldown Menu Location: Tools > Area
Keyboard Command: ptarea
Prerequisite: Set Area Defaults

Area by Closed Polylines

This command will calculate and report the area of single area and multiple area closed polylines. In the case of multiple areas, the user can choose to have the areas totaled (**Total Multiple Areas**) into a single result or to generate data for each area separately. Area by Closed Polyline will also automatically find special Carlson attributes attached to the polyline, in addition to capturing the area itself. These attributes will appear in the report, which can be the standard report or which can be presented in the Report Formatter, which itself links to Excel and Access. For example, property names and owner names, as applied to a polyline using the Mine modules, will report out automatically using Area by Closed Polyline. The command "Draw Lots from File..." will apply "extended entity data" to the lot polylines, which includes the lot name, and this will also report out when using Area by Closed Polyline. In addition, lot names, or any interior text whatsoever, can be captured and included in the report. The plot of the area on-screen can be canceled if only the report is desired.





Prompts

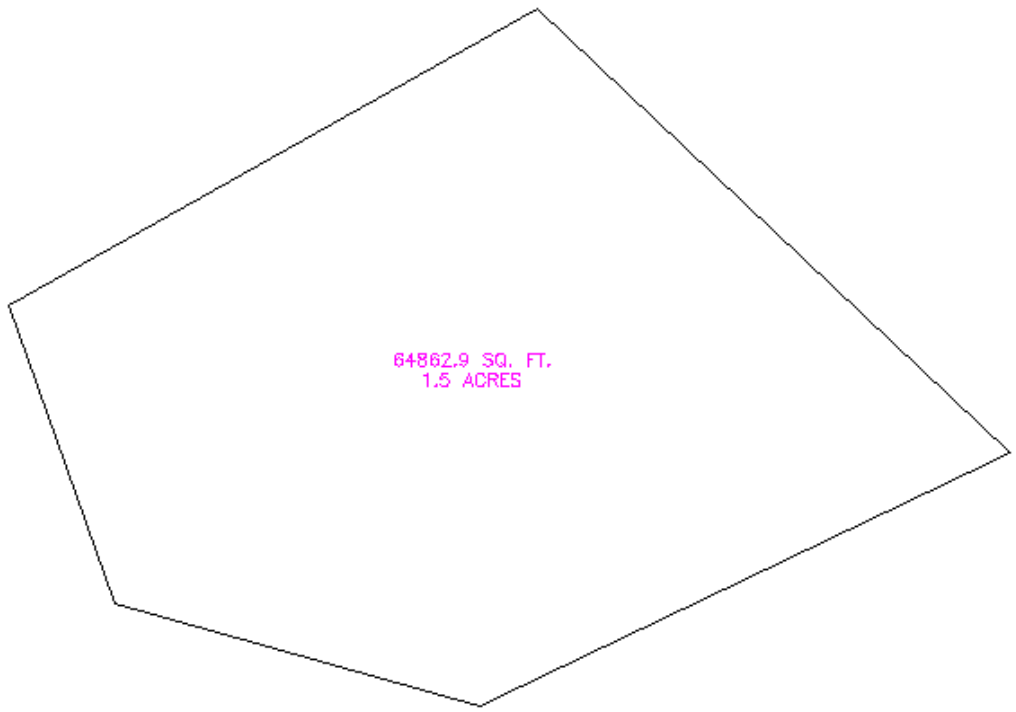
Select Area Polyline: *select the area polyline*
SQ. FEET: 64862.9 SQ. YARDS: 7207.0 SQ. MILES: 0.0
ACRES: 1.5 PERIMETER: 1018.7
Pick area label centering point (Enter for none): *pick a location*

When auto-placing labels at the rear of lots or when aligning labels by the sides of the lot the user will also be prompted to pick one or more centerlines (**Select the Centerline Polylines**). The routine will find the closest centerline and use this to determine the location of the front and back corners of the area.

When additional interior text is selected, the standard report will include that text:

Polyline Area 11/17/2004 12:49
Polyline Area: 43560.0 sq ft, 1.00 acres
Polyline Perimeter: 838.35 ft
Text: 16 Sf: 43560.0; Ac: 1.00

In this case, the "16" refers to Lot 16, and appears in the report because the lot number and existing area labeling were selected along with the polyline for the lot.



Pulldown Menu Location: Tools > Area
Keyboard Command: plarea
Prerequisite: Set Area Label Defaults

BearingBearing Intersect

This command locates a point at the intersection of two lines. The lines can be defined by picking two points, selecting a line or typing in a bearing. After the lines are defined a point symbol is located at the point of intersection. The figure below shows a graphical example of this command.

Prompts

[Enter] to use preview point/or select 1st Base point ?

Point number or pick point>: *[Enter]*

Pressing [Enter] causes the program to use the point coordinates highlighted by the preview arrow.

Define 1st angle by (Line/Points?Right/Azimuth/Bearing) <Bearing>: L

Select Line or Polyline that Defines 1st Bearing: (select line)

2nd Base point ?

Point number or pick point: *(pick point)*

Define 2nd bearing by (Line/Points/Right/Azimuth/Bearing) <Line>:

[Enter] to use preview point/or pick 1st point that defines 2nd bearing.

Pick point or Point number: *(pick point)*

2nd point that defines 2nd bearing ?

Pick point or Point number: *(pick point)*

The point is then located at the computed point of intersection.

Bearing-Bearing Intersect
Bearing-Distance Intersect

Pull-Down Menu Location: Tools, Locate at Intersect >
Prerequisite: Execute *Drawing Setup* to set Defaults.

BearingDistance Intersect

The Bearing-Distance Intersection command prompts the user for a base point from which the known bearing intersects. Then define the bearing by one of three methods. The bearing can be defined by picking two points, selecting a line with the same bearing or by typing in the bearing in the form of Qdd.mmss (similar to the Locate by Bearing command). Next the user is prompted for a base point from which the known distance radiates. After entering the known distance a circle is drawn radiating from the selected base point and a line defined by the bearing is extended to intersect the circle. The user then picks the correct point for the solution desired and a point symbol is located at the selected intersection. The command then erases the temporary circle and line.

NOTE: Except where noted, most of the Carlson routines leave the selection of the appropriate osnap mode up to the user. If a command turns on an osnap the prompt line of a command will notate which osnap is on by enclosing it in brackets. For example if the MIDpoint Osnap is on: [mid on] will appear in the point prompt line. Each of the predefined point symbols have a POINT entity at the center of the symbol, therefore the NODE ObjectSNAP mode should be used when snapping lines or other drawing entities to point symbols.

Prompts

[Enter] to use preview point/or select known Bearing base point ?

Pick point or Point number: *(pick point)*

Define bearing by (Line/Points/Bearing <P>: *L*

Select Line that defines Bearing: *(select line)*

Known distance base point ?

Pick point or Point number: *(pick point)*

Enter/Pick Distance: *40.41*

[int on] Pick Intersection point ([Enter] to cancel): *(pick point)*

Enter Point Number <55>: *[Enter]*

This prompt appears only if Automatic Point Numbering is set to No.

Enter Point Symbol Number <4>: *[Enter]*

This prompt appears only if point symbol prompting is set to Yes. Symbol number four is located at the computed coordinate and labeled point number 55.

Pull-Down Menu Location: Tools, Locate at Intersect >
Prerequisite: Run Drawing Setup to set defaults.

DistanceDistance Intersect

This command creates a point at the distance-distance intersection from two base points. The program prompts for two distances and two base points. The two possible intersections (A,B) are shown on the screen. You can either pick near the desired intersection or type in the letter A or B. The A intersection is clockwise from the first point.

Prompts

Select 1st base point

Pick point or point number: *1*

Points/<1st distance>: *46.72*

Select 2nd base point

Pick point or point number: *2*

Points/<2nd distance>: *38.96*

Pick near solution or Enter [A] or [B]: *pick a point*

Pull-Down Menu Location: Tools, Locate at Intersect>
Prerequisite: None

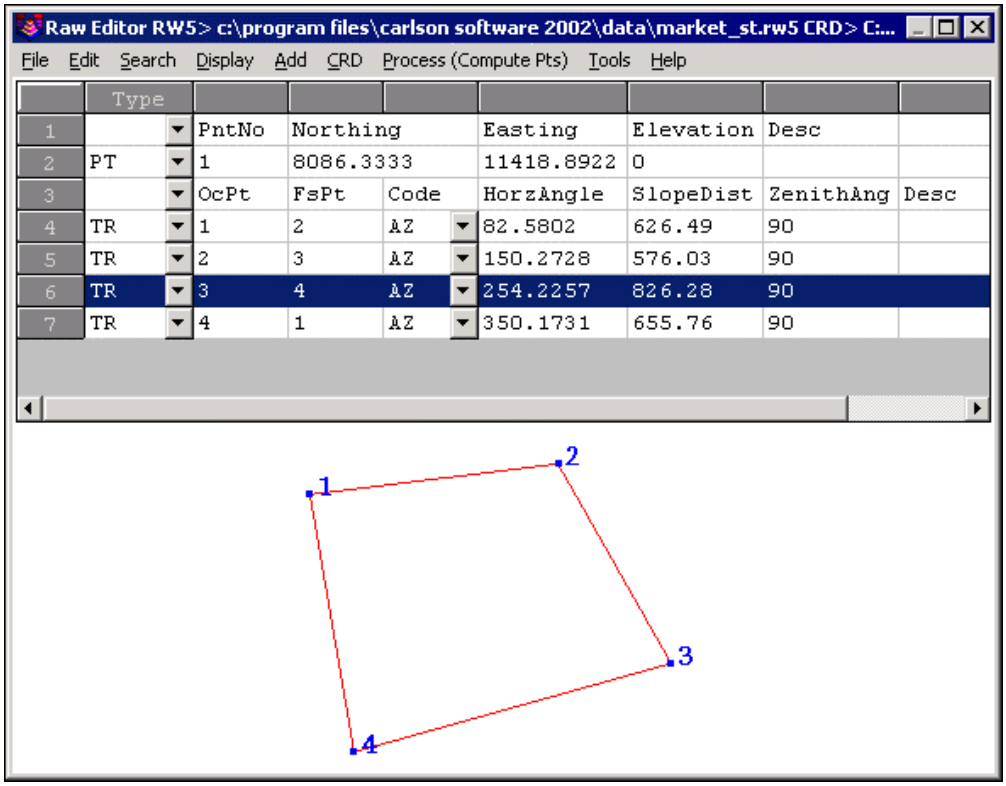
EditProcess Raw File

This program reads or creates a raw data (.RW5) file that contains various lines of data (records) that could be likened to a surveyor's field book. You can specify point coordinates, job information, notes, and the angles and distances that make up traverse or sideshots records. Once the raw data is created or read it can be processed/reduced to coordinates that are stored in the current coordinate (.CRD) file.

The raw file can also be created or appended using the Locate Point, Traverse, Sideshot, and Inverse commands on the Cogo menu. To store the data inputs from these commands into a raw file, first toggle on the Raw File ON/OFF command on the Cogo menu. It is possible to always have the raw data file open to store data inputs. To enable this option, choose Configure from the Settings menu, then choose Survey Settings, then choose General Settings. Turn on the Automatic Raw File toggle in this dialog.

The raw files created by TDS data collector programs are also compatible without conversion. The command Data Collectors on the Tools menu has options for reading other data collectors native file formats and converting them to raw data (.RW5) format. Within the raw data editor, the File menu includes an import menu for converting raw data from other formats.

When you select the Edit-Process Raw Data File command you are prompted to specify the name of the raw data (.RW5) file. The current coordinate file is used automatically. To change the current coordinate file, use the Set Coordinate File command in the Points menu before starting this command. If no coordinate file is current, the program will prompt you to set the current coordinate (.CRD) file.



Edit-Process Raw Data File uses an optional graphics window to display the points and traverse lines in real time. As data is entered or edited, the graphics window will be updated to show the configuration or new configuration of the traverse. The graphics window is toggled on or off from the Display — Graphics Window menu inside the raw file editor. Within the graphics window, real time zoom and pan are available. You may select whether to zoom or pan from the Display — Graphics Window menu also. To zoom in press and hold the left mouse button and drag in the direction of the + symbol. To zoom out, press and hold the left button and drag in the direction of the - symbol. To pan, set the graphics window to pan mode, then press and hold the left mouse button and then drag to desired position. With the Resize Text option on the text becomes smaller/larger in the view when you zoom in/out.

Edit-Process Raw Data File uses a spreadsheet for editing the raw data as shown. Each row of the spreadsheet represents one record of data. There are 14 types of data records. The type of data record is shown in the first column. Different record types use different numbers of columns. Whenever the data record type changes between rows, a record header is added to the spreadsheet that describes each column of data in the following row.

To edit the raw data, simply highlight the cell and type in the new value. To change the type of record, pick on the down arrow in the first column and choose a new data type from the list. To delete a row, highlight any cell in the row and hit the Delete key or choose Delete Row from the Edit menu. Records can be added pressing the Insert key,

pressing the down arrow key from the last line in the spreadsheet, or by choosing one of the add records from the Add menu.

Record Types

TR (Traverse)

The traverse record contains the occupied point number, foresight point number, angle mode, horizontal angle, distance, vertical angle and description. When processed, this record will calculate and store the coordinates for the foresight point. Traversing also moves the setup by making the traverse foresight point the next occupied point and the traverse occupied point becomes the next backsight point. The different angle modes are NE for northeast bearing, SE for southeast, SW for southwest, NW for northwest, AZ for azimuth, AL for angle left, AR for angle right, DL for deflection angle left and DR for deflection angle right. To set the angle code, pick on the Code down arrow and choose from the list. The horizontal and vertical angles should be entered as dd.mmss. For example, 45.2305 is 45 degrees, 23 minutes and 5 seconds. The vertical angle can be shown as vertical angle (0 degrees level), zenith angle (90 degrees level) or elevation difference. The vertical angle mode is set in the Display menu. The distance mode is also set in the Display menu as either slope or horizontal distance. The description field is used as the foresight point description.

SS (SideShot)

The sideshot record is the same as the traverse record except that sideshot does not move the setup.

HI (Instrument and Rod Height)

This record sets the instrument and rod heights used in elevation calculations. This record should precede any traverse and sideshot records that you want the heights applied to.

BK (BackSight)

The backsight record contains the occupied point number, backsight point number, backsight azimuth and the set azimuth. This record should precede any traverse and sideshot records that use this setup. If no backsight point is entered, the program uses the backsight azimuth to turn angles from. The Set Azimuth is the circle reading of the instrument when sighting the backsight. A Set Azimuth of zero is the default.

PT (Store Point)

The store point record consists of a point number, northing, easting, elevation and description. When processing, this data will be stored as a point in the coordinate file.

DS (Description)

The description record is an additional note appears in the spreadsheet editor and printouts. This record is not used in processing.

CL (Closing Shot)

The closing shot record is the traverse record where the foresight point is the closing point for the traverse. This record is used by the adjustment commands in the Process menu. There should be only one CL record in each Traverse loop (Name Record) in the raw file. If there is no CL record, the process adjustment routines will prompt for which shot is the closing shot.

AB (Angle Balance)

The Angle Balance is the traverse record that the Angle Balance routine in the Process menu uses to compare the angle between the occupied point and foresight point of this record with a user-specified reference angle. There

should be only one AB record in the raw file. If there is no AB record, then the Angle Balance routine will prompt for which shot to use as the angle balance.

CL + AB (Closing Shot and Angle Balance)

This record is used as both the closing shot and angle balance records.

FD (Foresight Direct)

The foresight direct is a traverse record used in a direct and reverse set. When the program finds one the of direct-reverse measurement records, it will look for the other three records to complete the set.

FR (Foresight Reverse)

The foresight reverse is a traverse record used in a direct and reverse set.

BD (Backsight Direct)

The backsight direct is a traverse record used in a direct and reverse set.

BR (Backsight Reverse)

The backsight reverse is a traverse record used in a direct and reverse set.

NAME (Traverse Name)

This record acts as an identifier for the group of records that make up a traverse. All the records after the NAME record belong to that traverse up to the next NAME record or the end of the file. This record allows you to have multiple traverses in one raw file. When running one of the Process commands, the program will bring up a list of all the traverse names. Simply choose which traverse to process. If you have only one traverse in the raw file, then you don't need the NAME record.

Menus

File

The File menu of the Raw Editor dialog box contains commands for printing, importing, exporting, and saving files.

- **Print**: This command displays the raw file data using the standard report viewer. You can print the report, draw it in the drawing, or save it to a file.

- **Import**: These routines convert raw data from other formats into the current Carlson Survey RW5 format. The converted raw data will be added to the end of any existing data in the editor. In many cases, the raw data file to import can be downloaded directly from the data collector or instrument using the Data Collectors command. The following supported formats (along with their standard file extension) are listed here.

- **Sokkia/Leitz (.RAW)**: Several data collectors generate this format including the SDR series collectors.

- **Wild/Leica (.GSI or .RAW)**: This reads the Leica raw file in Wildsoft, Liscad or 10-20-30-40 format. There is an option to specify for direct-reverse shot order if any.

- **SMI**: (.RAW)

- **Geodimeter**: (.OBS or .RAW)

- **Nikon**: (.RAW)

- **MDL Laser**: (.CDS)

- **Fieldbook (.FBK)**: From Softdesk or Land Development Desktop.

- **SurvCOGO**: (.RAW or .TXT)

- **PC Cogo**: (.BAT)
- **Survis**: (.RAW)
- **C&G**: (RAW)
- **Caice**: (.DMP)
- **Carlson**: (.RW5)
- **Maptech**: (.FLD)
- **Sokkia SDR**: (.SDR or .RAW)
- **TDS**: (.RW5)
- **3TA5**: (.TXT)

- **Export**: These routines convert the AgStar raw data (.RW5) file to other formats. The following file formats are supported.

- **Caice**: (.DMP)
- **Fieldbook**: (.FBK)
- **Sokkia**: (.SDR)
- **Save/Save As**: These commands save the raw data as a ∗.RW5 file.
- **Exit**: This command exits the raw file editor.

Edit

Under the Edit menu of the Raw Editor are basic cut, copy, and paste commands for editing. To edit the raw data, simply highlight the cell and type in the new value. To change the type of record, pick on the down arrow in the first column and choose a new data type from the list. To delete a row, highlight any cell in the row and press the Delete key, or choose Delete Row from the Edit menu. Data can be hidden from view with the Hide and Show commands.

Search

Under the Search menu of the Raw Editor are basic Find and Replace commands. You enter in a string you want to find and, optionally, what you want to replace it with.

Display

The Display menu of the Raw Editor contains settings to control how the information is displayed.

- **Vertical**: Vertical can be shown as Vertical Angle, Zenith Angle, Elevation Difference, or None. You pick the item off the menu list, and a check mark is placed in front of the selection.

- **Distance**: There are two choices for showing distance in the Raw Editor, slope and horizontal. You pick the item off the menu list, and a check mark is placed in front of the selection.

- **Graphics**: There are three toggles in this menu.
- **On/Off**: Toggle whether the graphics mode is available.
- **Zoom/Pan**: Toggles whether the cursor will zoom or pan
- **Text Resize**: Toggles whether text in the graphics screen should resize when zoom factor changes.

Add

You can add records by pressing the Insert key, pressing the down arrow key from the last line in the spreadsheet, or by choosing one of the add records from the Add menu of the Raw Editor. Record types are shown above in this section.

CRD

The CRD menu of the Raw Editor contains commands for manipulating points in a coordinate file.

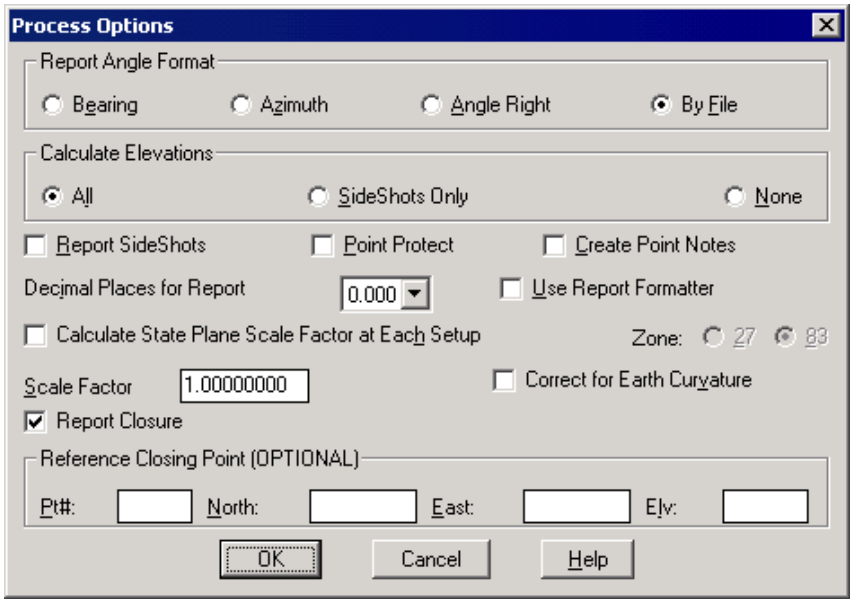
- **Edit Point**: This command creates new points in the coordinate file, or edits existing points.
- **List Points**: This command generates a list of points from the coordinate file. The dialog box gives the total number of points, and allows for listing a certain range of points. The list is displayed in the report viewer. See List Points in the Points chapter for more details.
- **Set Coordinate File**: Allows you to change the current coordinate (.CRD) file

Process (Compute Points)

The commands under the Process (Compute Points) menu of the Raw File Editor provide various methods for processing the raw file and for storing the calculated points in the coordinate file. The different types are described separately for each adjustment.

No Adjust

This command processes the raw file and stores the calculated coordinates to the coordinate file. The name No Adjust means that no angle balance or traverse adjustment will be applied.

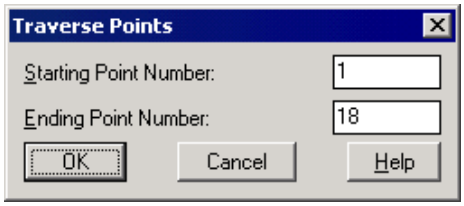


The dialog box prompts for the following options.

- **Direct-Reverse Vertical Angles**: For any direct-reverse raw data, you can choose to process the direct-reverse shots using only the foresight direct shot, or to balance all shots.
- **Report Angle Format**: This setting specifies the angle format for the report. The By File option specifies that the report should use the angle format in the RW5 file.
- **Calculate Elevations**: The Calculate Elevations option controls for which points elevations will be calculated. For example, if the traverse point elevations have already been adjusted and you need to recalculate the sideshot elevations, then use the SideShots Only option.
- **Report SideShots**: This option specifies whether to include the sideshot data in the process results report.
- **Point Protect**: This option will check the coordinate file for existing point data before processing. If the foresight point number for any traverse or sideshot record is already a stored coordinate in the coordinate file, then the program shows a list of conflicting point numbers. You can either continue processing and overwrite the coordinate file coordinates with the calculated raw file coordinates, or you can cancel the processing to go back to the editor to change foresight numbers.

- **Create Point Notes**: This option will generate a note file (.NOT) named after the coordinate file. The note file contains additional descriptions for points. With this option active, the text from all note records (DS records) will be stored to the note file for the foresight point number preceding the note records.
- **Decimal Places for Report**: Controls the number of decimals used in the report, ranging from zero to four.
- **Use Report Formatter**: Allows you to customize the process results report by selecting the layout of the fields to display. The report formatter can also output the report to Microsoft Excel and Access. Without the report formatter, the program generates a standard results report.
- **Calculate State Plane Scale Factor at Each Setup**: This will calculate a scale factor for each TR and SS record. This scale factor is calculated as the average of the scale factors at the occupied and foresights points. At these points, the scale factor is calculated as the state plane grid factor minus the elevation divided by the earth radius [SF = Grid Factor - (Elev/Earth Radius)]. In order to calculate the state plane scale factors, the traverse coordinates must be in state plane coordinates. When this option is selected, the program will prompt for the state plane zone to use. Select either Zone 27 or 83.
- **Scale Factor**: This factor is multiplied by the slope distance for the traverse and sideshot records.
- **Correct for Earth Curvature**: This adjusts the calculated points for the effect of the Earth's curvature. Typically this adjustment is small and adjusts the elevation more than the horizontal.
- **Reference Closing Point (OPTIONAL)**: This is an optional field for entering the coordinates to compare the ending traverse point with. This reference closing point is used to calculate the closure. If you do not specify this option the command will use the starting coordinate as the reference closing point.

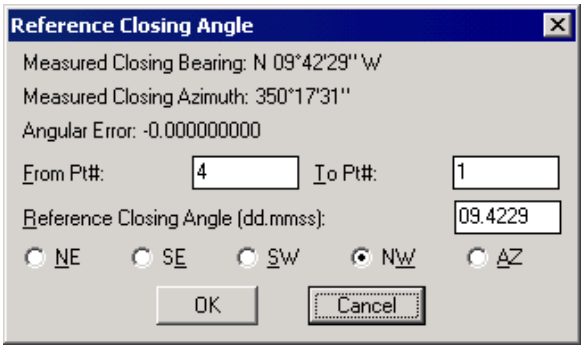
After you select OK for the first dialog box, the Traverse Points dialog box appears. Enter the starting and ending point numbers.



The command reads the raw file to set the defaults for these point numbers, which are used to calculate the closure. The difference between the ending point and the reference closing point is the closure error. The sum of the traverse distances from the starting to the ending point is used as the total distance traversed.

After you pick OK for the second dialog box, the program starts processing the raw file from the top record down. The result is displayed in the standard report viewer, which you can use to save, print, or draw the report.

Angle Balance

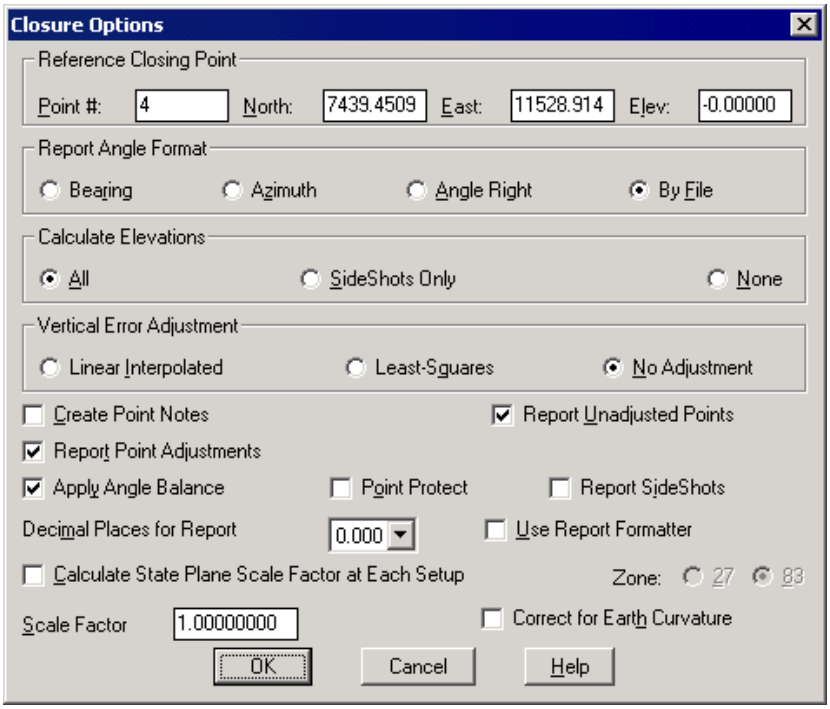


This process command applies an angle balance to the traverse lines when calculating the coordinates. This command displays the same dialog box as No Adjust. The angle balance takes the angular error divided by the number of traverse lines and adjusts the angle of each traverse line by that amount. The angular error is the difference between

the angle balance shot and a reference angle. The angle balance shot is specified as a type AB or CL+AB record in the raw file. If no AB record is found in the raw file, then the command prompts for which traverse shot to use as the angle balance shot. The angle from the angle balance shot is calculated as the angle from the occupied point to the foresight point. The reference angle can be specified as a bearing, as an azimuth, or by two point numbers in the dialog box shown. The angle balance report shows the unadjusted points, the unadjusted closure, the angular error, the adjusted points, and the adjusted closure. Typically, but not always, applying the angle balance correction improves the traverse closure.

Compass, Crandall, Transit

These process commands apply the selected rule of Compass, Crandall, or Transit, to the traverse lines when calculating the coordinates. The dialog box is the same for all three methods.



- **Reference Closing Point**: This is the field for entering the point number and coordinates to compare the ending traverse point with. The reference closing point is used to calculate the closure.
- **Direct-Reverse Vertical Angles**: For any direct-reverse raw data, you can choose to process the direct-reverse shots using only the foresight direct shot, or to balance all shots.
- **Report Angle Format**: This setting specifies the angle format for the report. The By File option specifies that the report should use the angle format in the RW5 file.
- **Calculate Elevations**: The Calculate Elevations option controls for which points elevations will be calculated. For example, if the traverse point elevations have already been adjusted and you need to recalculate the sideshot elevations, then use the SideShots Only option.
- **Create Point Notes**: This option will generate a note file (.NOT) named after the coordinate file. The note file contains additional descriptions for points. With this option active, the text from all note records (DS records) will be stored to the note file for the foresight point number preceding the note records.
- **Report Unadjusted Points**: Includes the unadjusted points in the report.
- **Vertical Error Adjustment**: Adjusts the vertical error for elevation difference.
- **Report Point Adjustments**: Reports the adjustment of each point.
- **Apply Angle Balance**: Applies the Angle Balance method to the traverse lines for calculating coordinates.

- **Point Protect**: This option will check the coordinate file for existing point data before processing. If the foresight point number for any traverse or sideshot record is already a stored coordinate in the coordinate file, then the program shows a list of conflicting point numbers. You can either continue processing and overwrite the coordinate file coordinates with the calculated raw file coordinates, or you can cancel the processing to go back to the editor to change foresight numbers.
- **Report SideShots**: This option specifies whether to include the sideshot data in the process results report.
- **Decimal Places for Report**: Controls the number of decimals used in the report, ranging from zero to four.
- **Use Report Formatter**: Allows you to customize the process results report by selecting the layout of the fields to display. The report formatter can also output the report to Microsoft Excel and Access. Without the report formatter, the program generates a standard results report.
- **Calculate State Plane Scale Factor at Each Setup**: This will calculate a scale factor for each TR and SS record. This scale factor is calculated as the average of the scale factors at the occupied and foresights points. At these points, the scale factor is calculated as the state plane grid factor minus the elevation divided by the earth radius [SF = Grid Factor - (Elev/Earth Radius)]. In order to calculate the state plane scale factors, the traverse coordinates must be in state plane coordinates. When this option is selected, the program will prompt for the state plane zone to use. Select either Zone 27 or 83.
- **Scale Factor**: This factor is multiplied by the slope distance for the traverse and sideshot records.
- **Correct for Earth Curvature**: This adjusts the calculated points for the effect of the Earth's curvature. Typically this adjustment is small and adjusts the elevation more than the horizontal.

After the command adjusts the traverse, it also recalculates the sideshots. The closure error is calculated as the difference between the closing shot and a reference point. The closing shot is specified as a type CL or CL+AB record in the raw file. If no CL record is found in the raw file, then the command prompts for which traverse shot to use as the closing shot. The foresight point is used as the closing coordinate. The reference point can be specified by point number or by entering the northing, easting, and elevation. The process results report shows the unadjusted points, closure error, adjustments to each traverse point, and adjusted points.

Tools

The Tools menu of the Raw Editor contains commands to help you manage points and angles in the raw file.

- **Direct-Reverse Report**: This command creates a report of direct and reverse shots along with the resulting averaged shots. The residuals are the difference between the measurement and the final average. A sample report is shown below.

```
Observations
Type Setup  FSight HorzAngle Distance  Vertical
BD   2      1      359.5950  173.8240  89.5308
BR   2      1      180.0000  173.8310  270.0654
FR   2      3      84.3443   176.9730  269.2822
FD   2      3      264.3439  177.0150  90.3142

BD   2      1      359.5953  173.8290  89.4920
BR   2      1      179.5957  173.8500  270.1059
FR   2      3      84.3536   177.0080  269.3238
FD   2      3      264.3524  177.0190  90.2722

Reduced Sets
HorzAngle Residual  FS Diff. BK Diff.
264.3446  0.0024    0.0004   0.0010
264.3535  0.0024    0.0012   0.0004
Vertical  Residual  Diff.    Distance  Residual  Diff.
90.3140   0.0209    0.0004   176.9940  0.0098    0.0420
90.2722   0.0209    0.0000   177.0135  0.0097    0.0110

Means
HorzAngle  SD       Distance  SD       Vertical  SD
264.3510   0.0024   177.0038  0.0098   90.2931   0.0209
```

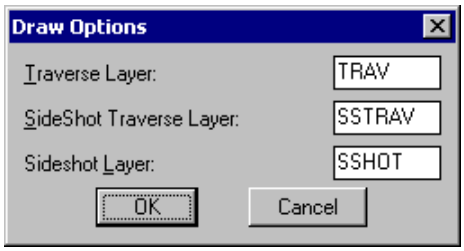
- **Reduce Direct-Reverse**: This command processes the direct and reverse shots and simplifies the raw file by replacing the sets of direct and reverse shots with the resulting average traverse record.

- **Update Raw from Points**: This command updates the horizontal angle, distance, and vertical angle data in the raw file using the coordinates from the coordinate file. This command can be used to generate a raw file from point numbers only. To create raw data from points, first create rows of Traverse and/or SideShot records in the spreadsheet containing only the occupied and foresight point numbers. Set the angle code to the desired format, then run Update Raw from Points. All the angles and distances will be filled in.

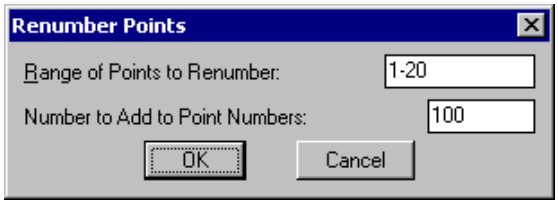
- **Find Bad Angle**: This command applies the angular error to each traverse record one at a time. The adjusted traverse record that improves the closure the most is reported as the Bad Angle. The angular error is the difference between the angle balance shot and a reference angle.

- **Append Another Raw File**: This command prompts for a raw file (.RW5). The file is read and its data is added to the end of the existing raw file. For example, if you are editing the raw file from a first day's work, and you have a separate raw file containing a second day's work, you can use this routine to add the second raw data file to the first.

- **Draw Traverse-Sideshot Lines**: This command draws lines for all the traverse and sideshot records. Sideshot Traverses are traverses that do not lead to the closing or ending point. You can specify different layers so that you can draw different lines with different colors. This command does not process the raw file. Instead it reads the raw file, and, for each traverse and sideshot record, it looks up the coordinates for the occupied and foresight points in the CRD file. You may need to run the No Adjust command under Process (Compute Points) menu of the Raw Editor before you run this command.



- **Renumber Points**: This command renumbers points in the raw file. This applies to all point numbers including: TR, SS, and PT records. You specify the range of point numbers to change and the amount to change in the dialog box.



Format of the Raw File (.RW5 extension)

The following are supported record header codes with their field headers.

BK Backsight

OP Occupy Point Number

BP Backsight Point Number (if 0 the next field's azimuth will be used for)

BS Back Azimuth

BC Back Circle

DS Description

LS Line of Sight

HI Height of Instrument

HR Height of Rod/Target

SP Store Point

PN Point Number

N North Coordinate

E East Coordinate

EL Elevation

- Point Description/Note

TR Traverse

SS Side Shot

CL Closure Record

AB Angle Balance Record

OP Occupy Point Number

FP Foresight Point Number

(one of the following 6)

AZ Azimuth (angle code 5)

BR Bearing (angle code 1 = NE, 2 = SE, 3 = SW, 4 = NW)

AR Angle Right (angle code 7)

AL Angle Left (angle code 6)

DR Deflection Angle Right (angle code 9)

DL Deflection Angle Left (angle code 8)

(one of the following 3)

ZE Zenith Angle (90 degrees level)

VA Vertical Angle (0 degrees level)

CE Change/Difference in Elevation from Instrument Point

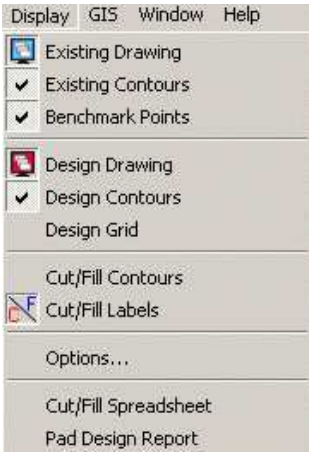
SD Slope Distance (if ZE or VA above)

HD Horizontal Distance (if CE above)

Menu Location: Tools
Prerequisite: None
Keyboard Command: RAWEDIT

Display Commands 10

Display



Check or uncheck the following options under the *Display* menu to show or hide elements from the drawing:

Existing Drawing: Shows/Hides all surveyed points and lines.

Existing Contours: Shows/Hides the contours lines for the existing field.

Benchmark Points: Shows/Hides the benchmark points.

Design Drawing: Shows/Hides the design arrows created in *Design Field*.

Design Contours: Shows/Hides the contours lines for the proposed field.

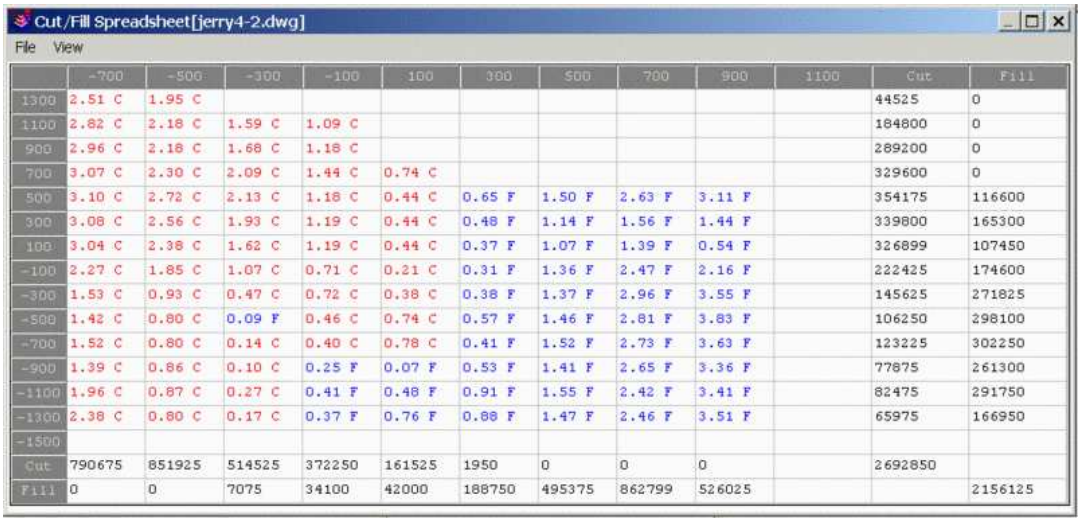
Design Grid: Shows/hides the grid created in the *Design Field* routine.

Cutfill Contours: Shows/Hides the cut/fill contour lines.

Cutfill Labels: Shows/hides cut/fill labels.

CutFill Spreadsheet

After designing the field, select *Cut/Fill Spreadsheet* from the *Display* menu to view a spreadsheet of the resulting field. If there are multiple subdivisions, you will be prompted to choose the subdivision you would like displayed. If you want the entire field displayed, press ESCAPE. A cut/fill spreadsheet will be displayed, with cut areas in red, fill areas in blue, and no-grade areas in green.

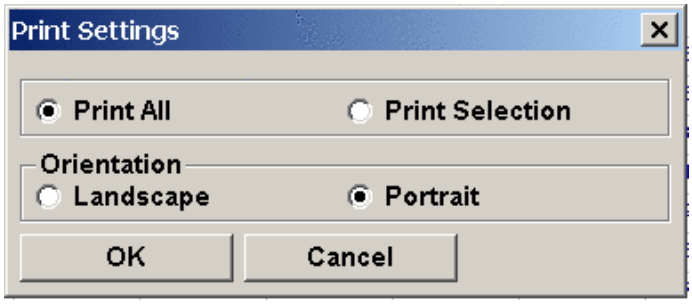

Cut/Fill Spreadsheet[jerry4-2.dwg]

File View

	-700	-500	-300	-100	100	300	500	700	900	1100	Cut	Fill
1300	2.51 C	1.95 C									44525	0
1100	2.82 C	2.18 C	1.59 C	1.09 C							184800	0
900	2.96 C	2.18 C	1.68 C	1.18 C							289200	0
700	3.07 C	2.30 C	2.09 C	1.44 C	0.74 C						329600	0
500	3.10 C	2.72 C	2.13 C	1.18 C	0.44 C	0.65 F	1.50 F	2.63 F	3.11 F		354175	116600
300	3.08 C	2.56 C	1.93 C	1.19 C	0.44 C	0.48 F	1.14 F	1.56 F	1.44 F		339800	165300
100	3.04 C	2.38 C	1.62 C	1.19 C	0.44 C	0.37 F	1.07 F	1.39 F	0.54 F		326899	107450
-100	2.27 C	1.85 C	1.07 C	0.71 C	0.21 C	0.31 F	1.36 F	2.47 F	2.16 F		222425	174600
-300	1.53 C	0.93 C	0.47 C	0.72 C	0.38 C	0.38 F	1.37 F	2.96 F	3.55 F		145625	271825
-500	1.42 C	0.80 C	0.09 F	0.46 C	0.74 C	0.57 F	1.46 F	2.81 F	3.83 F		106250	298100
-700	1.52 C	0.80 C	0.14 C	0.40 C	0.78 C	0.41 F	1.52 F	2.73 F	3.63 F		123225	302250
-900	1.39 C	0.86 C	0.10 C	0.25 F	0.07 F	0.53 F	1.41 F	2.65 F	3.36 F		77875	261300
-1100	1.96 C	0.87 C	0.27 C	0.41 F	0.48 F	0.91 F	1.55 F	2.42 F	3.41 F		82475	291750
-1300	2.38 C	0.80 C	0.17 C	0.37 F	0.76 F	0.88 F	1.47 F	2.46 F	3.51 F		65975	166950
-1500												
Cut	790675	851925	514525	372250	161525	1950	0	0	0		2692850	
Fill	0	0	7075	34100	42000	188750	495375	862799	526025			2156125

The following options are available:

Print: If you only wish to print part of the cutsheet, drag your mouse over the desired region while left-clicking the mouse. The desired region should now be highlighted. Select *Print* from the *File* menu.

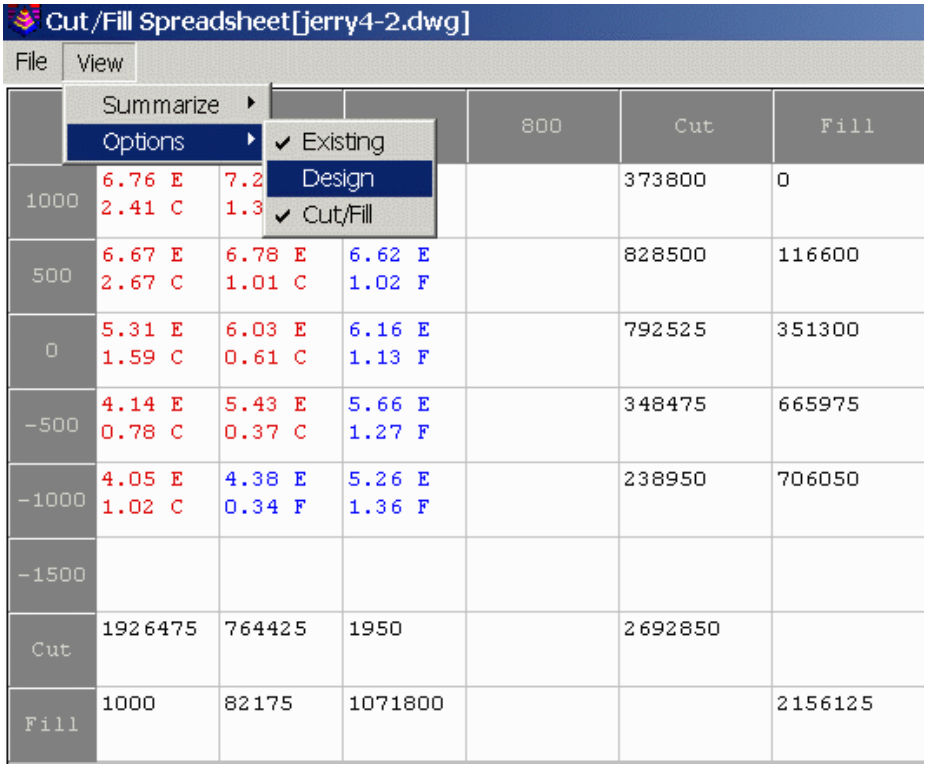


To print the selected region, check the *Print Selection* toggle. Other select the *Print All* toggle. You can also choose between *Portrait* and *Landscape* printing orientations. Click *Ok* to print.

Summarize: You may increase the grid cell size by going to *Summarize* under the *View* menu, and choosing a desired scale factor. Cutsheet grid sizes must be a multiple of the grid size chosen during the *Make Existing Ground Grid* routine.

	-700	-200	300	800	Cut	Fill
1000	6.76 E 4.36 P 2.41 C	7.22 E 5.88 P 1.34 C			373800	0
500	6.67 E 3.99 P 2.67 C	6.78 E 5.77 P 1.01 C	6.62 E 7.64 P 1.02 F		828500	116600
0	5.31 E 3.72 P 1.59 C	6.03 E 5.42 P 0.61 C	6.16 E 7.30 P 1.13 F		792525	351300
-500	4.14 E 3.36 P 0.78 C	5.43 E 5.05 P 0.37 C	5.66 E 6.93 P 1.27 F		348475	665975
-1000	4.05 E 3.03 P 1.02 C	4.38 E 4.72 P 0.34 F	5.26 E 6.62 P 1.36 F		238950	706050
-1500						
Cut	1926475	764425	1950		2692850	
Fill	1000	82175	1071800			2156125

Options: By default, only cuts and fills are displayed in the spreadsheet. However, the existing and design elevations of each cell can also be displayed by going to *Options* under the *View* menu.



Font Size: You may increase the font size by going to *Font Size* under the *View* menu, and select the desired font size. This feature is useful if you need to fit more data onto a printout.

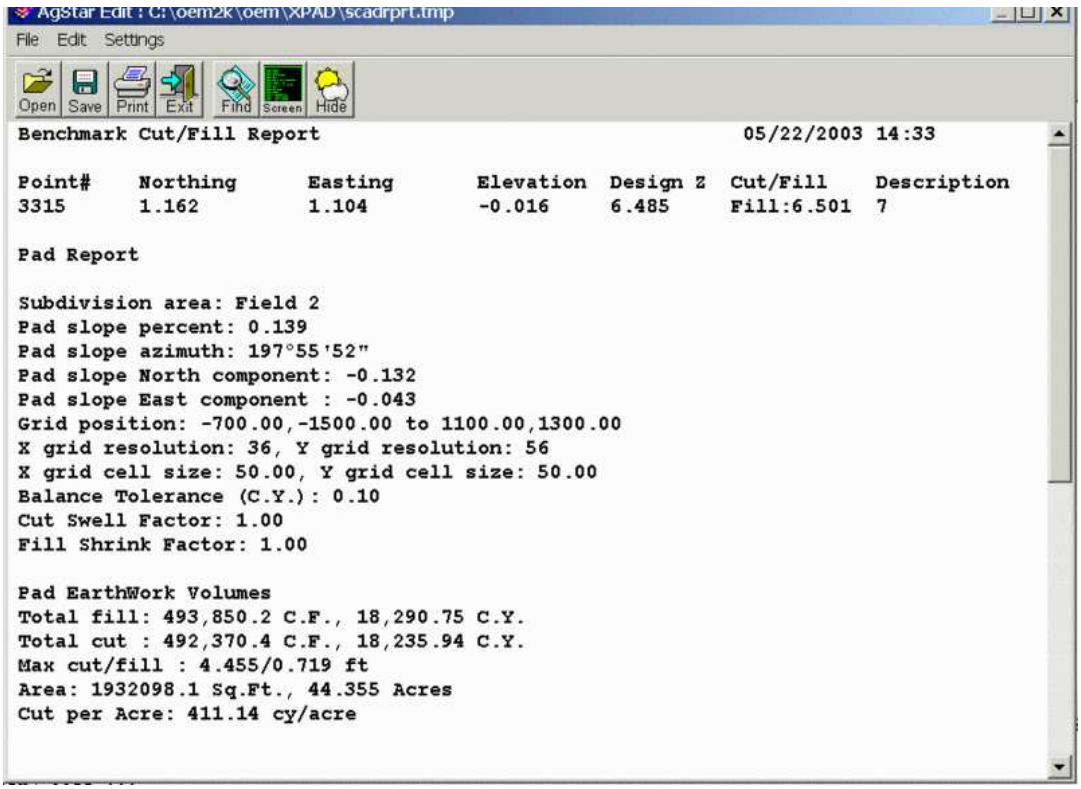
Import/Export: You can export/import the cutsheet to/from Excel or any other database program, through standard cut & paste procedures.

Quit: To quit, choose *Quit* from the *File* menu.

Pull-Down Menu Location: Display
Prerequisite: A field design
Keyboard Command: show_cutsheet

Pad Design Report

Select Pad Design Report from the Display menu to show the design and benchmark information for each field.



Pull-Down Menu Location: Display
Prerequisite: A field design
Keyboard Command: bmark_cf_report

Display Options

These settings apply to various AgStar commands.

Point Size: Sets the size of surveyed points in the drawing.

Benchmark Size: Sets the size of benchmark points in the drawing.

Surface Inspector Label Size: Sets the text size of labels created by Surface Inspector.

Cut/Fill Label Interval: Sets how many grid cells to skip between labels. Set this value lower to increase the density of the cut/fill labels.

Label Existing Surface Elevation: Controls whether to label the existing elevation along with the cut/fill labels.

Label Design Surface Elevation: Controls whether to label the design elevation along with the cut/fill labels.

Contour Interval: The elevation difference between each contour line.

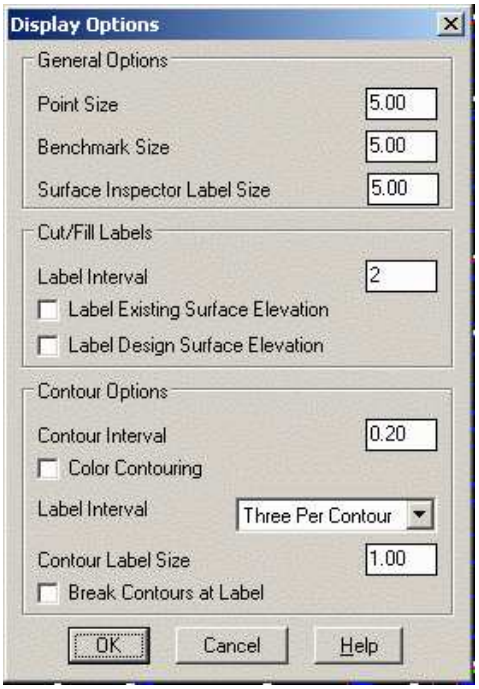
Color Contouring: Check this option to show the contour lines with a color gradient.

Label Interval: Controls how many elevation labels per contour.

Contour Interval: The elevation difference between each contour line.

Contour Label Size: Sets the text size for the contour labels.

Break Contours at Label: Controls whether to create a gap in the contours under the labels.



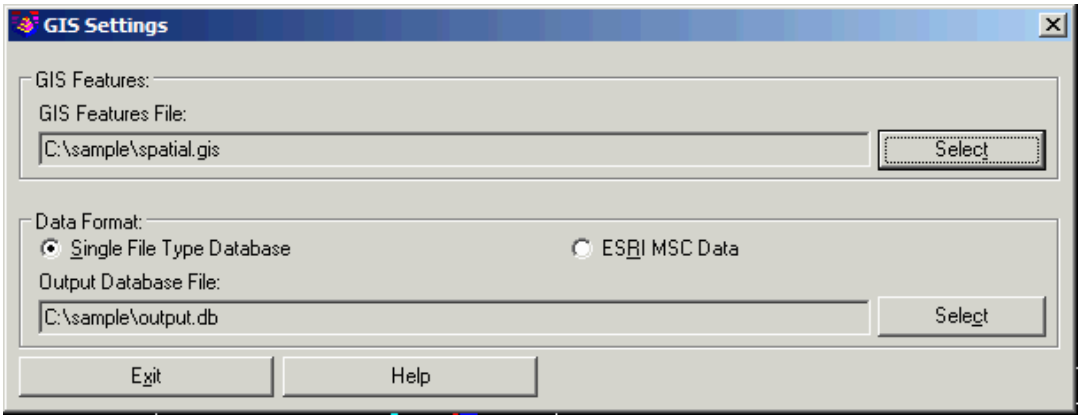
Pull-Down Menu Location: Display
Prerequisite: None
Keyboard Command: ag_display_options

GIS Commands 11

GIS Database Settings

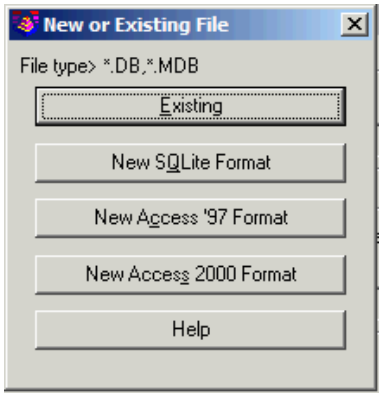
This command sets the current GIS Features and GIS Data Format. The GIS Features file (.GIS) defines the GIS features and the attributes for each feature. This file is set by the Define GIS Features command.

The Data Format defines where the GIS data will be stored. For Single File Type Database, the data is stored in an external database in either SQLite format (.DB) or MicroSoft® Access (.MDB). The Esri MSC Data stores the GIS data within the drawing file in a format that both Carlson and Esri use. Starting with ArcGIS 9.3, Esri added support for MSC which makes the DWG file a type of geodatabase with the feature definitions, GIS data and geometry all stored in the file.



Prompts

GIS Setting dialog Click both file buttons and select new or existing files.



Pulldown Menu Location: GIS
Keyboard Command: gis_config
Prerequisite: None

Define GIS Features

This command creates the Feature/Attribute data structure, or schema, for GIS functionality. The structure is stored in a special Carlson file with a (.GIS) file extension. A feature, such as a manhole, can have multiple attributes, such as Number of rungs, Type of material, Number of inlets, etc. Features can be organized into Categories: Utilities, Roads, Properties. The Category designation is an arbitrary way of organizing the features. Features and attributes can be imported from Field to Finish, Esri MSC data within the drawing, or from older Carlson Template Database

MDB files.

Features and attributes can of course also be defined "from scratch" in the Define GIS Features dialog box.

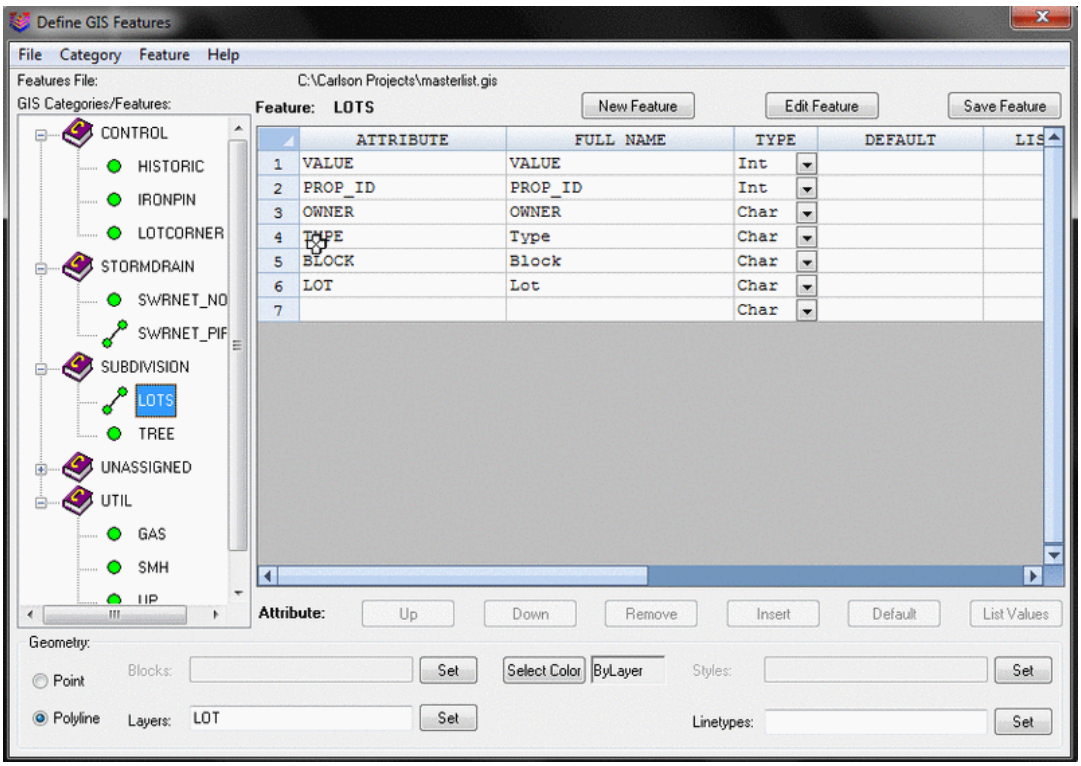
1) The first field to set is the GIS file you are working with. Use the File menu to create a new (.GIS) file or open an existing one for editing.

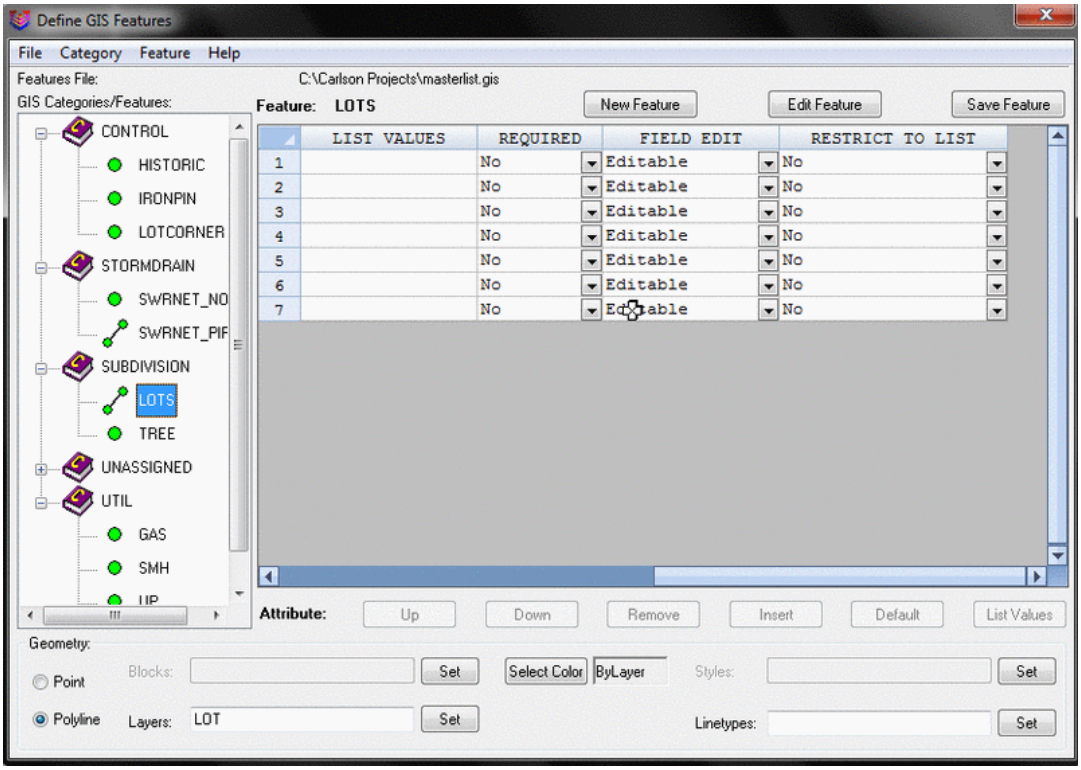
2) Next, set up one or more Category Names, using the Category menu. GIS feature codes can be categorized (e.g. STRUCTURES, UTILITIES, ROAD FEATURES, etc.). At least one category must be created.

3) Next, define Features, using the Feature menu. e.g. A category such as UTILITIES might have features such as manholes, light poles, fire hydrants, water valves, etc.

4) Lastly, define the attributes for the Feature. Each attribute has:

a) a Name
b) a Full Name, or Prompt
c) a Type - Integer, Character or Real
d) a Default Value - these can be preset, or read from a list of automatically generated values using the Default button
e) optionally, a List of values to pick from. Use the List Values button to build a list
f) whether the attribute is required
g) whether the attribute Value can be field Edited, appears as Read Only, or is Hidden
h) whether the value used is restricted to the list





Geometry Settings

At the bottom of the dialog box the user can specify the geometry settings for each feature, whether it is a line or point feature, what layer it is to be drawn on, what block to use to represent it, what text style to use, and what linetype to use.

Pulldown Menu Location: GIS
Keyboard Command: def_template
Prerequisite: None

Input-Edit GIS Data

This routine creates, reviews and appends GIS data linked to entities stored in the drawing.

The GIS Smart Prompting dialog has a spreadsheet format for editing the data fields. The GIS table to process is selected in the pull-down list in the upper right of the dialog. The GIS tables that are available depend on the tables that are defined in the current template database. Use the *GIS Database Settings* and *Define Template Database* commands to setup the tables. Once you select a table to process, the fields for that table are displayed in a spreadsheet format. If a field is related to a field in another table in the database, a "+" character is shown next to the field name. Picking the "+" will open another dialog box with the related data in the other table. The data in this related table is not editable, only the data in the initial linked table.

The bottom portion of the dialog has features for attaching images to the entity. Existing image files (BMP, JPG or GIF) can be linked by choosing the New option. The Update option will replace the current image with a newly selected image. The Delete option will remove the current, attached image. The Capture button will take a shot in the field using a configured camera and then attach the image to the entity. Different digital cameras can be used by picking Pick or Set Camera.

The *Input-Edit GIS Data* command is an excellent way to simply review the data associated with an entity. If the entity has GIS data, the banner line at the top of the dialog will display "Entity has GIS Data". If not, the banner

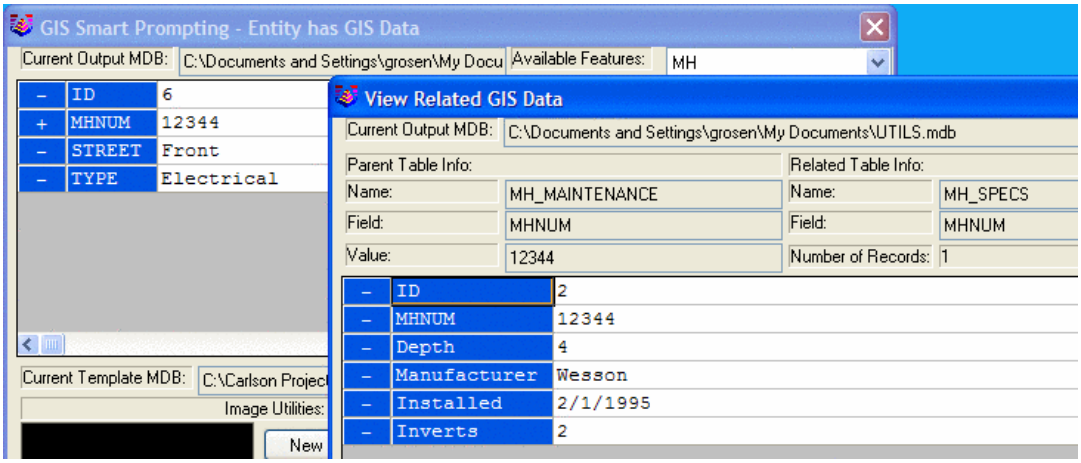
line will display "Entity has no GIS Data". Even when the entity has no data, the default values for the prompts will appear. Pressing OK will assign this data to the entity. To avoid assigning data to the entity (if it has none), press Cancel. Alternately, you can use the commands *GIS Inspector Settings*, followed by *GIS Data Inspector*, to review the data with no possibility of editing or inputting data in the process.

There are three methods for selecting the drawing entities to process: S for Select, P for Pick and N for Number:

Select Object method: With this method, you pick the drawing entity to process the data attached to that entity. When selecting a Carlson point, the point number is used to link to the database.
Pick method: For this method, you pick inside a closed polyline to process the data attached to that polyline.
Number method: Here you simply input the point number from the current CRD file to process.

Prompts

Select object (Number/Pick/<Select>): *P*
Pick a point inside polygon (Select/Number/<Pick>): *pick a point*
GIS Smart Prompting dialog *make selections*



Pulldown Menu Location: GIS
Keyboard Command: gisdata
Prerequisite: MDB GIS prompting must be created in *Define Template Database* and points or entities must exist to link GIS information to.

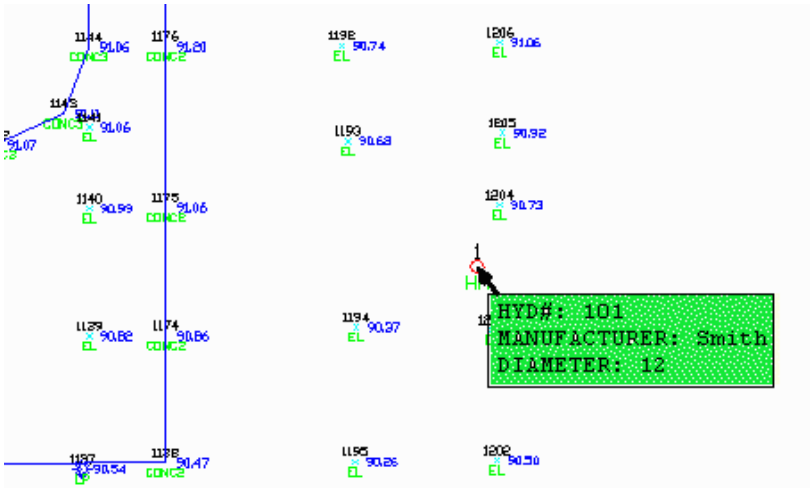
GIS Inspector

This command displays all or portions of the data attached to drawing entities in real-time. How much of the attached data is displayed is set by the command GIS Inspector Settings. When you move the cursor over an entity with GIS data, selected fields are displayed in a tooltip box next to the cursor. For data attached to closed polylines, you can move the cursor anywhere inside the polyline to show the data. Polylines that are closed will highlight with a solid fill as you inspect each one. Open polylines, such as road centerlines, will highlight with a solid fill generated along the length of the polyline. The solid fill color for all highlighting is set in GIS Inspector Settings.

The routine starts by prompting you to select entities. The entities that you select will be used by GIS Inspector. In the case of a large drawing, this selection allows you to limit the entities for inspector to a local area instead of having to process the whole drawing. Then after reading the entities, you can move the cursor around the drawing to inspect the GIS data. You can also use the arrow, page up and page down keys to pan and zoom the display. Pressing enter ends the routine.

Prompts

Select objects: *select entities with attached data*
Arrow keys=Pan; PageUp/Down=ZoomOut/In;
Zoom=Pick left-lower and right-upper corner;
Move pointer over entity with Gis Data (Enter to End): *move cursor over entities with data; press Enter to end*

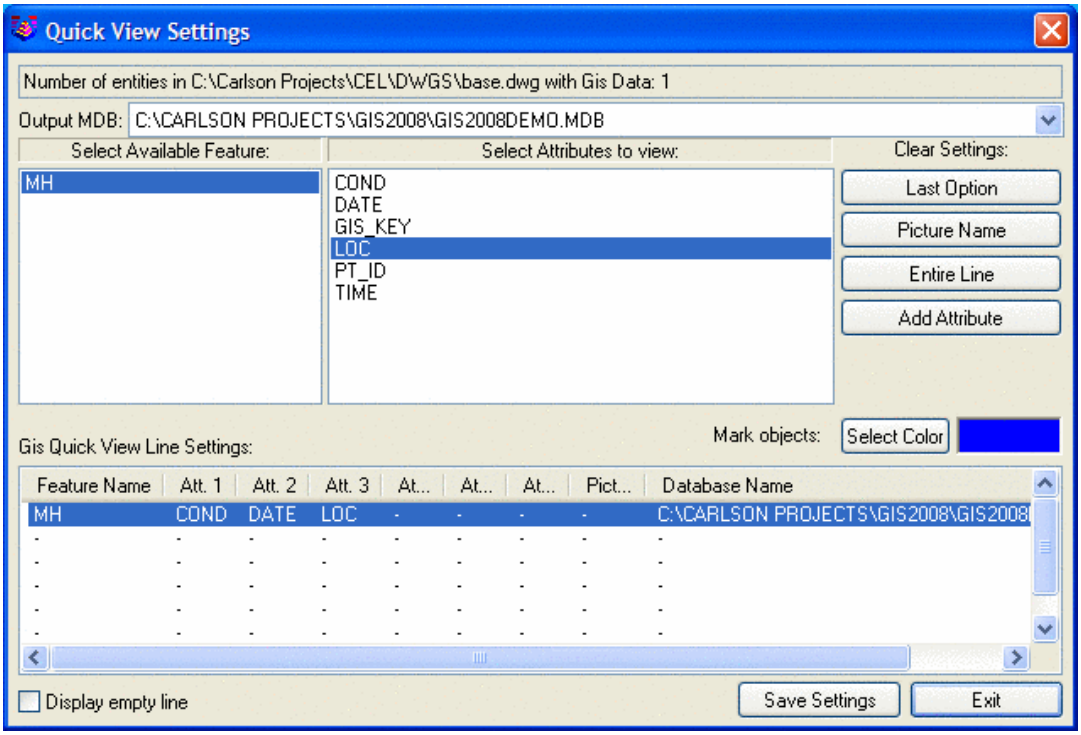


Pulldown Menu Location: GIS
Keyboard Command: gis_inspector
Prerequisite: MDB GIS Prompting must be created in *Define Template Database* and entities must have linked GIS information.

GIS Inspector Settings

This command sets up the fields to be displayed when using *GIS Data Inspector.* Each GIS table code can have different display options stored in the GIS Inspector Settings command.

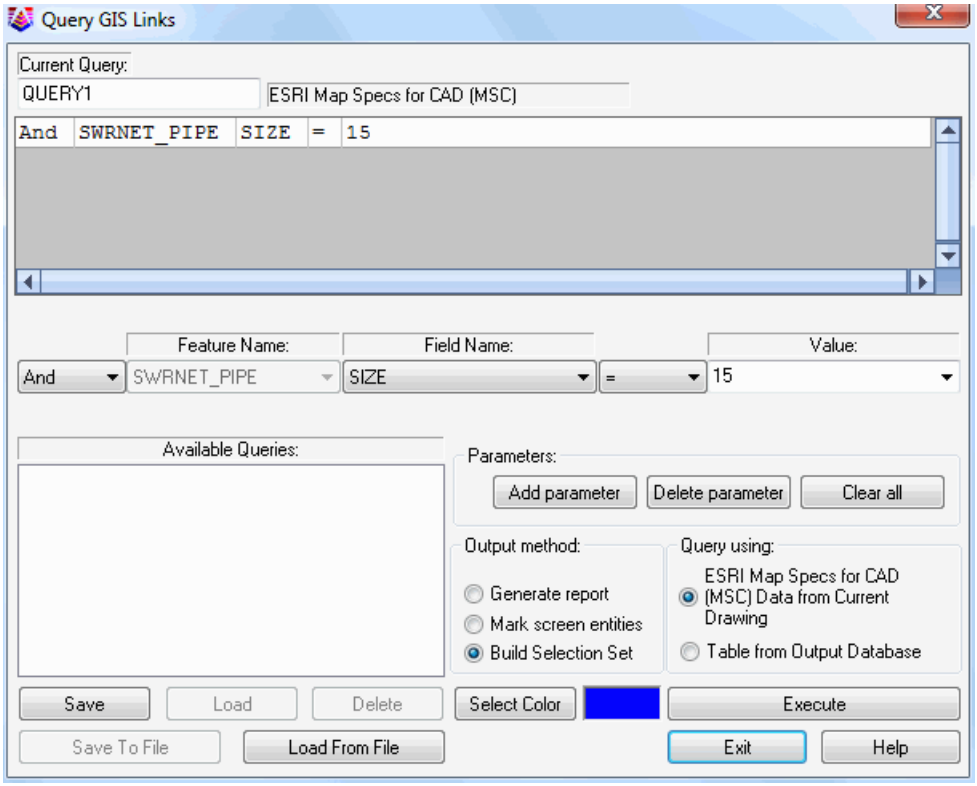
GIS Inspector Settings reads all the points and entities with GIS information currently linked in the drawing and displays a list of the linked data tables under the Available GIS Table column. When a GIS Table code is highlighted (i.e. 0001 or Road), the fields for this GIS table are displayed to the right in the Select Fields column. Up to 6 fields or lines of GIS data can be defined for display for each GIS code table, including one picture. To add a field to the display list, double-click on the field name. To remove a field from the display list, highlight the GIS table to remove from and then use the Clear Settings buttons. The Last Option button will remove the last field to display from the current GIS table. The Picture Name will remove the image from the display list. The Entire Line button removes all the fields from display for the current GIS table.



Pulldown Menu Location: GIS
Keyboard Command: set_inspector
Prerequisite: MDB GIS Prompting must be created in *Define Template Database* and points or entities must have linked GIS information.

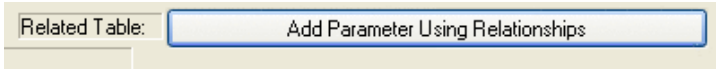
GIS Query/Report

This command applies a user-defined query on a data table or related tables with the database. Records in the table that pass the query can be reported or the associated entities can be highlighted in the drawing. The *Query Using* option in the main dialog box sets the source of the data table to process as either GIS data attached to selected drawing entities or from the current Output MDB file.



The query is defined in the dialog shown here. To add a query, enter a new query name in the in the space underneath Current Query. If there is already a name there, just highlight and type over it with a new name, then hit Clear All to clear out existing query lines and get full access to all Feature Names.

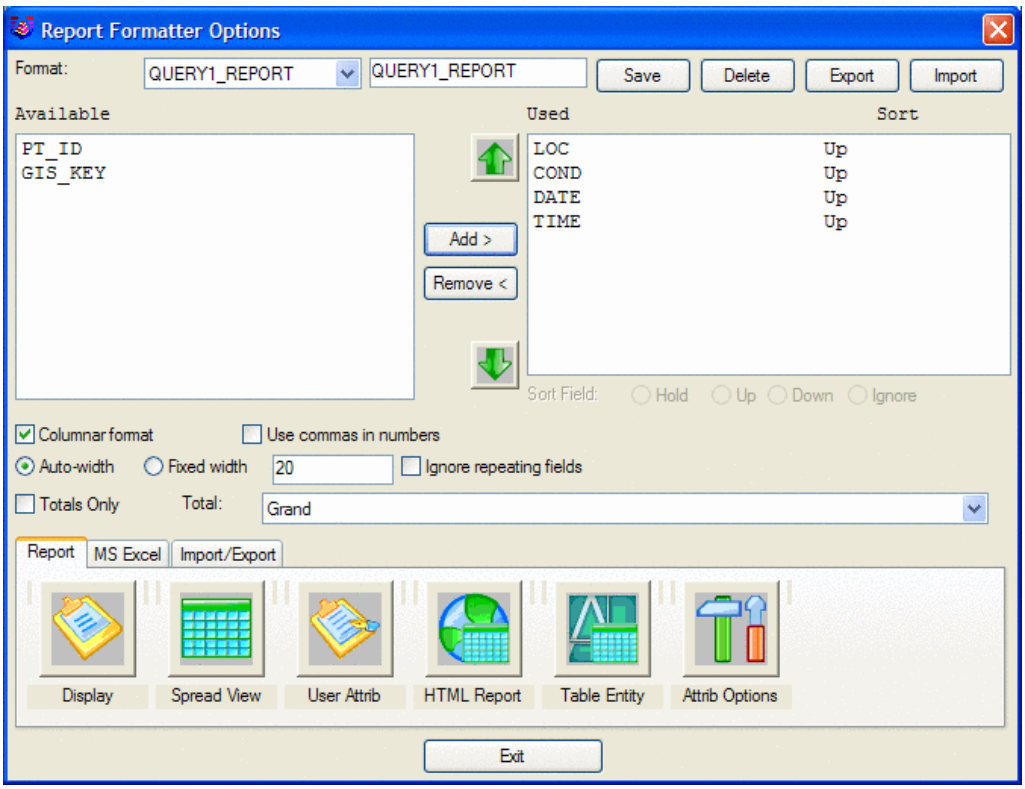
The top portion of the dialog contains a list of the query parameters. To add a parameter, select a Feature Name from the pop-up list. The available features will either be all the features found in the GIS links of the drawing or all the features from the Output MDB file depending on the *Query Using* option. Once the feature is specified, the Field Name pop-up list contains all the available fields in the feature. Choose a field from this list. Next choose the operator (=, >, etc.) from the operator list. The Value pop-up list contains all the different values for that field that are found in the current data set. You can either select one of these values or type in another value into this field. If a Field Name relates to another Feature, when you select that Field, an additional button will appear allowing you to add a query parameter from the related feature.



When all the parameter values are set, pick the Add Parameter button. Once a feature is selected and add a parameter is added, the Feature Names list becomes unavailable because any additional query parameters must come from that feature, or relate through that primary feature. When all the parameters are defined for the query, you can save these settings by filling out a name Current Query field and then picking the Save button. This query can be recalled later by highlighting the query name and clicking the Load button. The Delete button removes the highlighted query. The Save, Load and Delete functions operate on the current set of queries active in the program. The Save To File and Load From File functions read and write the collection of queries to a .QRY file for managing different sets of queries and sharing with others.

Pick the Execute button to process the query. The Mark Screen Entities option will set the color of entities with GIS data that match the query to the specified color. The Build Selection Set option creates a selection set of the entities that pass the query. To use this selection set in other commands, enter "P" for previous at the "Select objects:" prompt. With the Generate Report option, the program will bring up the Report Formatter which allows you to choose the fields to include in the report and the report format. If the Highlight Screen Entities option is on, then

the program will highlight the entities with GIS data that pass the query. Point entities are highlighted by drawing a box around the point and polylines are highlighted by solid fill. Shown here is the report for all manholes with a Condition of Good.



Pulldown Menu Location: GIS
Keyboard Command: gis_query
Prerequisite: MDB file with data or entities with linked GIS information

Import SHP File

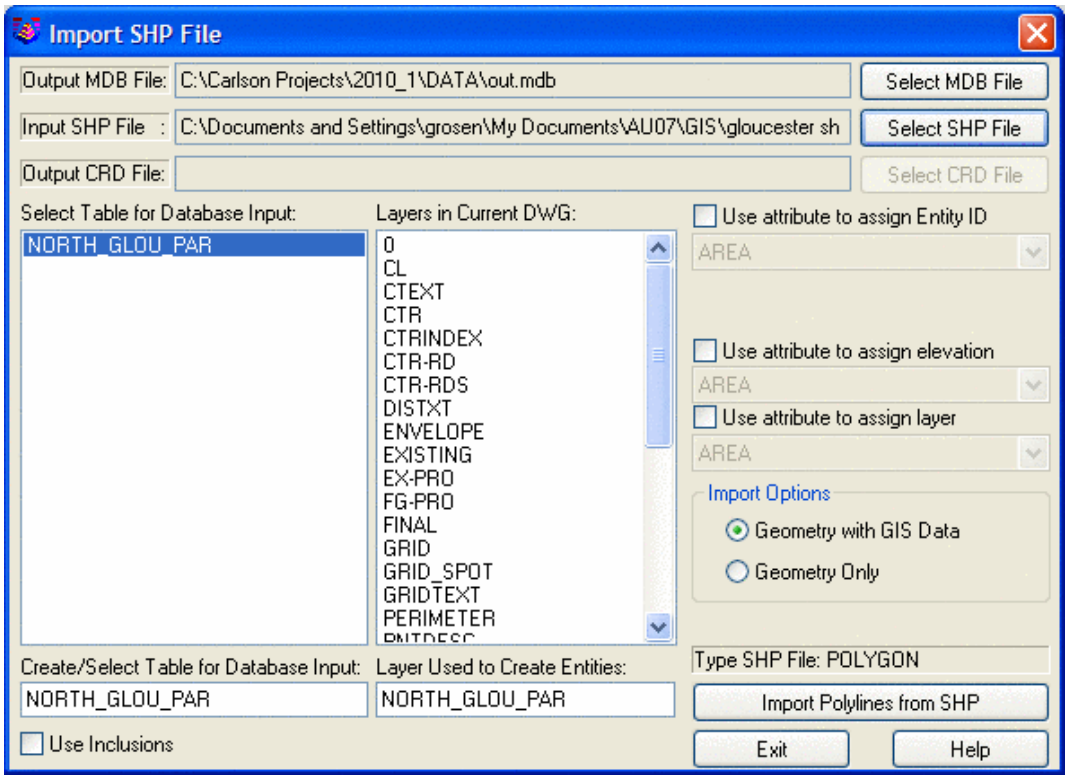
The Import SHP File command converts ESRI SHP files into Carlson drawing entities and can also optionally write the available attribute data to an external Access MDB file and create GIS links between the drawing entities and the records in the database. Use the Geometry with GIS Data Import Option to accomplish this. Use the Geometry Only Import Option to just draw the linework. If you don't need the data, this option is much faster.

The Import SHP File dialog displays the Output MDB file to add data to and the source SHP file to be imported. SHP files are similar to entities in one layer in CAD. You must specify the table name to store the data in the MDB database and the layer name for the entities to be created. Typically these names are the same or near equivalent as the SHP file name. Once these names are entered, the Import Polylines from SHP button becomes available. Pick this button to import the SHP files entities and database. You can also assign elevations by a specified data attribute.

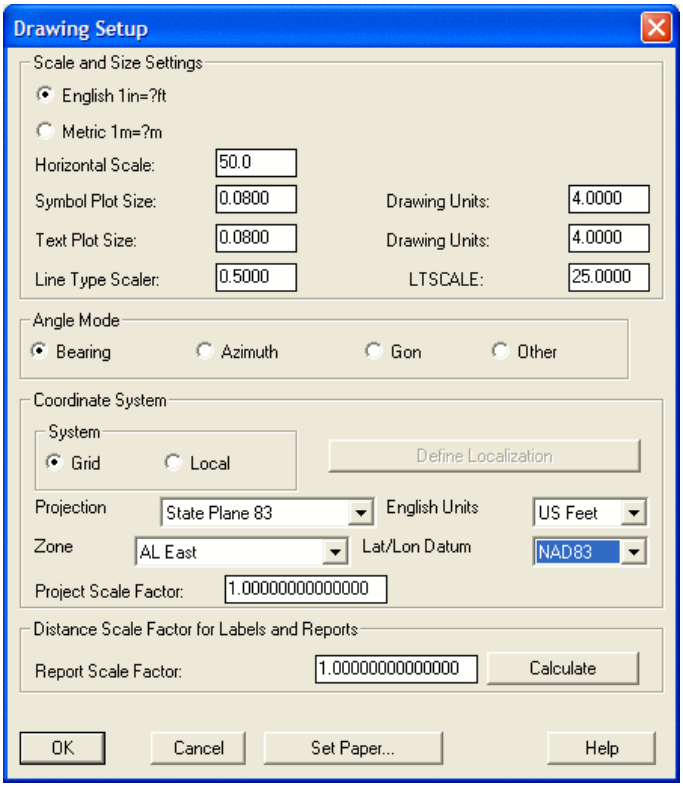
There are primarily three types of ESRI SHP files: Points, Arcs and Polygons. Each will provide different options on Import. Once the SHP file is selected, Carlson detects the data contents of the file and sets the dialog options for importing either polygons, arcs or points. Carlson GIS also supports the use of three other types of SHP files: PointM, PolylineM and PolylgonM.

Both Arc and Polygon SHP files are brought into Carlson as polylines in the drawing, with attribute data stored in an external Access .MDB database file if that option is selected.

Point SHP files are imported in a three step process. The first step uses the Import SHP File command to create a coordinate file (.crd) for the points in the SHP file and a corresponding table in the output MDB file for the points database. The second is to use Draw Locate Points to draw the points from the CRD file into the drawing. The third step uses Create Links to select the points in the drawing and link the database to these plotted points.



Note: If the SHP file you are Importing is in a different Projection or Units than that specified in the Drawing Setup, then a transformation will occur during Import, as long as the (.PRJ) Projection file is present with the SHP set of files. If there is no (.PRJ) file with the SHP, then no transformations will occur.



Pulldown Menu Location: GIS
Keyboard Command: import_shp

Export SHP File

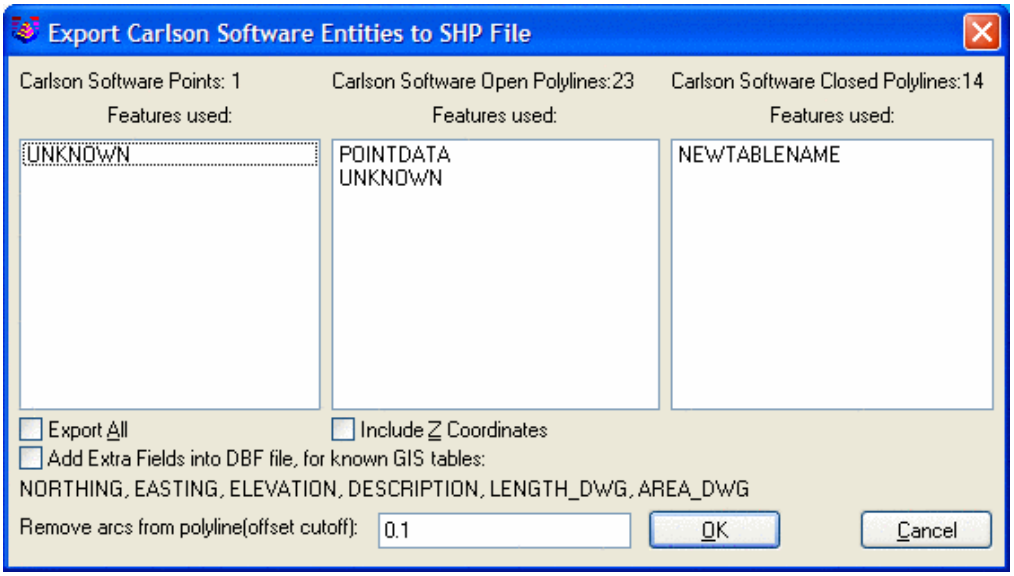
This command creates a SHP file from the selected entities in the drawing. After selecting entities to be converted, a dialog shows the number of Points, Polylines (Arcs) and Closed Polylines (Polygons) found in the drawing selection set. Those Points, Arcs and Polygons with database information linked are displayed with their database table names. Any Points, Arcs and Polygons without linked database information display as unknown.

Highlight the Point, Arc and Polygon tables to output or selects Export All to select all entities including the UNKOWN entities to export into SHP files. The Export SHP File commands outputs all entities selected into SHP files with the same name as their table name into a subdirectory selected. Also Points can be stored in the ESRI Arcview database as 3D X, Y and Z coordinates when Include Z Coordinates is toggled on. SHP files do not have arc entities. So the export routine will convert arcs and polyline arcs into a series of small chords segments. The Offset Cutoff field sets the maximum horizontal shift allowed between the original arc and the chord segments.

These SHP files can be imported into ESRI's Arcview product. Database GIS links in Carlson are converted to SHP files by storing the GIS database information into DBF files for ESRI's Arcview product to read and link to.

Prompts

Specify Name for SHP File dialog *select .SHP file name*
Select objects *select entities*
Export Carlson Entities to SHP File dialog *choose settings, click OK*



Pulldown Menu Location: GIS
Keyboard Command: export_shp
Prerequisite: None

Help Commands 12

OnLine Help

This command opens the AgStar on-line Help File.

Menu Location: Help
Prerequisite: None
Keyboard Command: [F1] or HELP

Training Movies

This command opens an application that lets you choose from several training movies. The movies provide instruction for all aspects of AgStar.

Menu Location: Help
Prerequisite: None
Keyboard Command: RUN_MOVIES

Index

plus-0.5in
.PRO, 27
2D Polyline, 62
3 Point Curve, 68
3D Polyline, 63, 67, 191, 192
3D Viewer Window, 55

Add, 22, 31, 88, 99, 100, 138, 199, 202, 234, 236, 256
Add Polyline Vertex, 48
Adjust Field Elevation, 151
Align, 32, 39, 65, 113, 137–141, 145, 146, 213, 225
Align GPS To Local Coordinates, 137
Align to Benchmark, 140, 141, 146
Alphanumeric, 102, 212
Angle Balance, 234, 235, 238, 239, 242
Angle Mode, 78, 104, 105
Append Another Raw File, 241
Arc, 34, 68, 175, 220, 250, 257, 259
Area, 24, 113, 145, 152, 163–165, 208, 226, 227, 229, 231
Area by Closed Polylines, 229
Area by Interior Point, 229
Area Defaults, 223, 227–229
Area Name Inspector, 153
ASE, 36, 126–128
Assign Subdivision Area Name, 152
Attributes, 191, 206, 219
AuthorizingAgstar, 12
Automatic Point Numbers, 168, 171
Azimuth, 43, 78, 102, 104, 105, 113, 140, 169, 221, 234, 241, 242

Barscale, 75
Bearing, 40, 42, 43, 68, 78, 104, 220–222, 231, 232, 242
BearingBearing Intersect, 231
BearingDistance Intersect, 231
Benchmark, 2, 140, 141, 144, 247
Boundary, 162, 229
Break, 41, 49
Break at Intersection, 41
Break by Closed Polyline, 40, 49
Breakline, 199

Centerline, 31
Change, 12, 26, 42, 86
Change Elevations, 42, 212, 215
Change Layer, 57
Change Properties, 41
Circle, 64, 89, 204, 241
Close, 21, 33, 34, 40, 49, 106, 124, 177, 178, 229, 259
Close Polylines, 48
Closure, 242
Command Entry, 1, 9
Configuration, 88, 118, 125, 128, 136
Configure, 101, 104, 110, 121, 124–128, 130, 131, 136, 139–141, 227, 233
Configure Survey, 110, 119–123, 129, 132–134, 137, 140, 141, 169, 212
Contours, 244
Convert, 45
Coordinate File, 116, 176, 199, 208
CooRDinate File Utilities, 212
Coordinate File Utilities, 212
Coordinate Transformation, 213
Copy, 37, 91, 199
Create, 30, 32, 65, 100, 159, 202
Create Design Surface Points, 217
Create Existing Ground Grid, 149
CSI GBX Pro, 118
Custom, 6, 7, 16, 25, 96, 192, 194
Custom Linework Label Formatter, 73
Cut/Fill, 150, 244
CutFill Spreadsheet, 244
Cut Sheet, 105
Cutsheet, 245

D, 2–4, 7, 9–11, 13–17, 21, 22, 25–34, 36–40, 42–52, 55–60, 62–67, 70–72, 74, 80–102, 104–108, 112–117, 123–128, 130, 131, 134, 135, 138–147, 149–155, 158–161, 163–165, 167–174, 176, 178, 179, 181–218, 220–225, 227, 228, 230–239, 241, 242, 246–248, 250–259
Database, 22, 102, 117, 250, 259
Data Collecto, 235
Data Files in DWG Directory, 101
Decimal Places, 106
Defaults, 74, 107, 111, 231
Define GIS Features, 250, 251
Definition, 175, 176, 188
Delete Layer, 36
Delta Angle, 107
density, 149, 247
Design, 2, 3, 80, 96, 105, 149, 151–155, 161, 199, 244
Design Commands, 148
Design Field, 149, 154, 166, 167, 244
Discussion Groups, 17
Display, 244, 246–248

Display Commands, 243
Display Options, 247
DistanceDistance Intersect, 232
Distance MeasureSwathing, 158
Distance Measure with Pen, 160
Dozer, 29
Draw, 11, 20–22, 24, 34, 39, 62–69, 71–76, 78–80, 83, 85, 88, 89, 94, 96, 99, 103, 115, 150, 153, 160, 163, 165–167, 171–174, 176, 179, 193, 196, 197, 199, 202–204, 212, 215, 218, 225, 244
Draw Barscale, 75
Draw Commands, 61
Draw Field to Finish, 172, 173, 175, 176, 178–180, 188, 189, 193, 199
Drawing Inspector, 59, 60
Drawing Setup, 27, 29, 34, 72, 78, 101–103, 129, 132, 172, 194, 223, 224, 231, 232, 258
DrawLocate Points, 169
Draw North Arrow, 75, 76
Draw Subdivision Line, 151

Eagle Point, 177, 178
Earth Curvature, 117, 238, 240
Edit, 3, 23, 36–39, 41, 42, 44, 45, 51, 52, 56, 62, 75, 84–86, 125, 127, 128, 176, 185, 188, 189, 195, 199, 202, 203, 224, 235–237, 240, 241, 251
Edit Commands, 35
Edit Point, 216, 237
Edit Point Attributes, 218
Edit Points, 208
Edit Polyline Vertex, 48
EditProcess Raw File, 232
Edit Symbol Library, 100
Edit Text, 44
Elevation, 33, 42, 59, 63, 104, 121, 126, 130, 136, 138, 139, 151, 157, 167–171, 192, 193, 217, 223, 237, 239, 242
Elevation Difference, 104, 117
Enter and Assign, 171
EnterAssign Point, 223
Entities to Polylines, 46
Equipment, 2, 3, 110, 121, 123–125, 130, 136
Equipment Commands, 118
Erase, 36, 79, 171, 212, 215, 219, 225, 228
Erase Point Attributes, 219
Erase Points, 209
Exit, 34, 86, 100, 121, 124, 125, 131, 136, 137, 141, 143–147, 158–160, 236
Explode, 37, 38, 65
Export Google Earth File, 33
Export LandXML File, 31
Export SHP File, 259
Export TextASCII File, 210
Extend, 39, 40, 83
Extend by Distance, 39, 40, 62
Extend To Edge, 39
Extrapolate, 167

Factor, 127, 204
Field to Finish, 11, 115, 169, 170, 173, 176, 178, 218
File Commands, 19
Fillet, 45
Find Bad Angle, 241
Follow Elevation, 157
Freeze Layer, 58

Geodimeter, 118, 119, 210, 215, 235
GIS Commands, 249
GIS Database Settings, 250, 252
GIS File, 117, 169
GIS Inspector, 253
GIS Inspector Settings, 253, 254
GIS Query/Report, 255
Grid, 2, 116, 238, 240, 244
Grid File Utilities, 30

Header, 212
Help Commands, 261

Image Adjust, 51
Image Clip, 50
Image Frame, 51
Import/Export Topcon TIN File, 29
Import/Export Trimble TTM File, 29
Import Google Earth File, 28
Import LandXML File, 26
Import SHP File, 257, 258
Import TextASCII File, 209
Impulse Laser, 120
Info, 13
Input-Edit GIS Data, 202, 252
Input-Edit Point, 208
Inquiry, 104
InRoads, 187
Insert, 64, 65, 67, 70, 80, 83, 177, 195, 208, 233, 236
Insert Symbols, 66
Installing Agstar, 4
Interior Point, 227
Inverse, 220–222, 229
Inverse with Area, 227, 228
Isolate Layer, 58, 170

Join, 45, 184
Join Nearest, 44–46

Label, 104, 107, 108, 115, 154, 168, 169, 175, 206, 220, 223, 225, 226, 231, 244, 247, 248

Label Angle, 108
Label Area Names, 154
Label Elevations, 115
Label Zeros, 171, 175
Layer, 28, 33, 41, 57, 59, 63, 72, 94, 106, 107, 163, 169–171, 173, 175, 179, 191, 192, 205, 225
Layer ID, 57
Layers, 28, 36, 91, 115, 169, 192
least squares, 113, 213
Leica, 118, 121, 122
Leica GPS System 500, 120
Leica TC Series, 122
Length, 68, 80, 111
License Agreement, 4, 13, 14
Line, 3, 22, 25, 28, 36, 41, 46, 62, 63, 74, 78, 89, 96, 97, 102, 103, 106–108, 122, 144, 151, 163, 174, 175, 178, 179, 184, 189, 194, 198–200, 210, 212, 214, 221, 222, 227, 229, 231, 232, 241, 254
List, 58, 69, 84, 88, 91, 105, 116, 175, 179, 208, 210, 215, 251
List Points, 208, 215, 237
Locate on Real Z Axis, 104, 115, 168, 170
Locate Point, 11, 171
Locate Within Coordinate Range, 170

Make, 4, 7, 88, 110, 124, 126, 128, 130, 136, 170, 185–187, 245
Manual Total Station, 123
Maptech, 236
MicroStation, 29, 30
Monitor GPS Position, 146
Move, 36, 39, 40, 62, 88, 110, 151, 212, 215, 219, 223, 254
Move Point Attributes, 219

NADCON, 213
National Geodetic Survey, 113
Navcom Configuration Guide, 123
Navcom GPS Setup, 125
Nearest Found, 173, 184
New, 20, 23, 43, 50, 89, 92, 101, 125, 140, 169, 173, 252
Nikon, 118, 129, 235
Nikon Total Stations, 129
Nodes, 170
Note File, 115, 117
Numeric, 102

Object Linking, 102
Object Snap, 81, 82, 96, 98, 171
Occupy Point, 220, 241, 242
Offset, 38, 45, 49, 103, 105, 106, 113, 114, 182, 202, 225, 259
Offset 3D Polyline, 45
Offset Cutoff, 30
OLE objects, 22, 92–94
OnLine Help, 262
Open, 21, 40, 48, 55, 73, 91, 131, 136, 253
Open Polylines, 49
Options, 20, 25, 66, 73, 87, 89, 91–95, 103, 105, 107, 108, 115, 118, 163, 172, 176, 192, 204, 212, 220, 221, 226–228, 245

Pad Design Report, 246
Pan, 55, 56, 135, 236, 254
Parameters, 193
PC PT Radius Point, 68
PC Radius Chord, 68
PDMODE, 102
PDSIZE, 102
Place Google Earth Image, 71, 72
Place Image by World File, 71
Plot, 22–26, 78, 88, 91–94, 96, 102, 103, 214
Point Defaults, 167, 169, 172, 223
Point Description, 117, 242
Point Entity, 200
Point Layer, 102, 191
Point Notes, 114, 115, 175, 238, 239
Point Number Format, 212
Point Protect, 26, 28, 32, 210, 217, 221, 222, 237, 240
Points, 11, 28, 33, 34, 42, 74, 102, 106, 115, 116, 168, 169, 171, 172, 174–177, 179, 187, 192, 195, 199–201, 207–213, 215–217, 219, 232, 233, 241, 244, 257–259
Polyline, 30, 36, 37, 41, 46–50, 59, 60, 62, 90, 157, 170, 174, 227, 229, 230, 257
Polyline File, 30
Polyline Info, 59
Polylines, 30, 34, 40, 45, 230, 253, 257
Pond, 163
Precision, 32, 80, 225
Process, 3, 112, 165, 177, 178, 189, 202, 226, 233–235, 237, 241
Product Overview, 1, 2
Profile, 31, 32
Projects, 200, 203

Quickstart Guide, 2

Raster Image, 69
Reduce Polyline Vertices, 30, 47
Reduce Vertices, 103
Regen, 56, 96
Registration, 12
Remove, 37, 50, 67, 88, 100, 191, 213, 223
Remove Polyline Arcs, 49
Remove Polyline Segment, 49
Remove Polyline Vertex, 50
Removing, 49

Renumber Points, 215, 241
repackaged, vi
Report, 31, 60, 102, 105, 166, 176, 187, 207, 208, 216, 227, 229, 237–240, 256
Report Log, 101
Resize Point Attributes, 219
Restore, 86, 131
Restore Due North, 57
Restore Layer, 58
Reverse, 40, 47, 115, 235, 240, 241
Reverse Polyline, 47
Rotate, 42, 43, 56, 137, 193, 194, 212, 215, 216
Runoff, 162, 164, 167
Run Off Tracking, 167
RW, 11, 106, 112, 178, 232, 233, 235–237, 239, 241

Save, 10, 21, 22, 27, 70, 73, 74, 86, 88, 90, 91, 100, 101, 125, 139, 166, 173, 188, 227, 256
Save As, 21, 90, 173, 188, 236
Scale, 25, 38, 41, 56, 65, 71, 75, 78, 103, 106, 113, 114, 116, 140, 141, 191, 194, 204, 216, 219, 224, 226, 238, 240
Screen Pick Point, 171
Section Name, 88
Set Coordinate File, 212, 237
Set Data Directory, 11
Set Environment Variables, 84
Set Perimeter, 149
Settings Commands, 77
Shade, 56, 185
SHP File, 259
Shrink-Wrap Entities, 67
Side Shots, 222
Simulation GPS, 130
Slope, 150
Slope Distance, 105, 242
Smart Prompting, 252, 253
SMI, 235
Smooth Contours, 103
Smooth Polyline, 47, 198
Softdesk, 235
Sokkia, 130–132, 235, 236
Special, 179–181, 185
SQL, 250
SRVPNO, 104, 168, 192
Stack Label Arc, 107
Stakeout Design Surface, 147, 160
Standard, 87, 95
standard deviation, 135
Start Point Number, 168, 169, 171
Startup Wizard, 1, 10, 11, 101
State Plane, 2, 112, 113, 121, 131, 136, 137, 139
Step, 124, 125, 140–145, 147, 149–154, 157, 160
Surface, 2, 27, 46, 164, 182, 193, 247
Surface Entities, 47
Surface Inspector, 2, 154, 247
Surface Model, 167
SurvCE, 2, 179, 191, 202
SurvCOGO, 235
Survey Benchmark, 141, 144
Survey Commands, 109
Survey Interior Surface, 143
Survey Master Benchmark, 145
Surveyor, 80
Survey Perimeter, 142
Survey Subdivision line, 144
SurvStar, 138
Symbol Library, 66, 194
Symbol Size, 102, 103
System Requirements, 1, 3

TDS, 178, 233, 236
Technical Support, 1, 17, 125, 128
Template, 10, 20, 21, 47, 88, 117, 199
Template Database, 117, 250, 252–255
Text, 11, 22, 34, 44, 59, 60, 65, 78, 87, 90, 93, 102, 103, 105–108, 154, 170, 172, 175, 179, 191, 192, 195, 198, 201, 210–212, 224, 225, 230, 233, 236
Text EnlargeReduce, 44
Text Size, 44
Text Style, 44
TFW file, 71
Thaw Layer, 58
TIF file, 71
Title Block, 103
TO, 14–16, 47, 58, 85, 97–99, 118, 127, 129, 182
Toolbars, 86
Tools Commands, 156
Topcon Total Stations, 132
TR-, 64
Training Movies, 262
Transit, 239
Translate Points, 216
Traverse, 221–223, 233–235, 238, 241, 242
Triangulation, 27
Trim, 38, 118, 126
Trimble, 2, 29, 134–136, 178, 210
TS, 15–17, 66, 81, 100, 132, 135, 198
Twist, 102, 225
Twist Screen Line, 57
Twist Screen Standard, 56, 57
Twist Screen Surveyor, 57
Typical Alignment Scenarios, 140

Undo, 36, 38–40, 50, 62, 63, 214, 228
Units Control, 79, 104
Universal Transverse Mercator, 213

Update, 16, 42, 108, 212, 215, 241
Update Drawing from CRD File, 214

Vertical Angle, 104, 223, 236, 237, 239, 242
Vertical Angle Mode, 104, 169
View, 25, 26, 54–60, 69, 72, 73, 79, 102, 127, 138, 208, 245, 246
View Commands, 53
Volume, 149, 150

Watershed Analysis, 161–163, 165–167
Wild, 171, 210, 235
Wildsoft, 235
Write Polyline File, 29

Zeiss, 210
Zone Polylines, 30
Zoom Center, 54
Zoom Extents, 54
Zoom Previous, 54
Zoom - Window, 54
Zoom IN, 54
Zoom OUT, 54